# SECTION 7 OPERATION

**A SUPPLEMENT TO THE OPERATION MANUAL FOR THE**

# CINCINNATI CL-800 SERIES LASER SYSTEM

**Edited for CINCINNATI Laser Control Software Version 8.2**

CINCINNATI®

CINCINNATI INCORPORATED
CINCINNATI, OHIO 45211

EM-544 (R-07/10)

COPYRIGHT © 2010
CINCINNATI INCORPORATED

# SECTION 7: OPERATION CONTENTS

## INTRODUCTION

## HMI OVERVIEW


LASER SYSTEM DISPLAY WINDOW ..... 7-1
- MENU BAR ..... 7-2
- TOOL BAR ..... 7-2
- STATUS INDICATORS CONTROL BAR ..... 7-3
- PROCESS STATUS BAR ..... 7-3
- APPLICATION WORKSPACE ..... 7-3
- OPERATOR CONSOLE CONTROL BAR ..... 7-3
- PROGRAM MODES CONTROL BAR ..... 7-4
- PROGRAM STATUS BAR ..... 7-4
- MAIN STATUS BAR ..... 7-4

FILE TYPES ..... 7-4
- NC PROGRAM FILE ..... 7-5
- PROCESS PARAMETER FILE ..... 7-5
- BATCH PROGRAM FILE ..... 7-6

CNC RUN WINDOW ..... 7-7


## CONTROL BARS


OPERATOR CONSOLE ..... 7-9
- RESONATOR CONTROLS ..... 7-9
- OVERRIDE SETTINGS ..... 7-10
- OPERATING MODES ..... 7-11

OVERRIDE ADJUST ..... 7-12
AUXILIARY FUNCTIONS ..... 7-13
PROGRAM MODES ..... 7-14
- PROGRAM TEST MODE ..... 7-14
- REWIND ..... 7-14
- BLOCK DELETE MODE ..... 7-15
- OPTIONAL STOP MODE ..... 7-15
- SINGLE BLOCK MODE ..... 7-15
- TRACE MODE ..... 7-15


## MENU COMMANDS


FILE MENU ..... 7-17
- NEW ..... 7-17
- OPEN ..... 7-17
- QUICK FILE OPEN ..... 7-19
- OPEN LOADED PROGRAM ..... 7-19
- OPEN CURRENT PROCESS ..... 7-19
- CLOSE ..... 7-20
- SAVE ..... 7-20
- SAVE AS ..... 7-20
- LOAD NC PROGRAM ..... 7-21
- LOAD CURRENT PROGRAM ..... 7-22
- LOAD CURRENT BATCH ..... 7-22
- PLOT NC PROGRAMS ..... 7-22
- RESTART LOADED PROGRAM ..... 7-24
- PRINT ..... 7-25
- PRINT PREVIEW ..... 7-25
- PRINT SETUP ..... 7-26
- MRU FILE LIST ..... 7-26
- OPEN OPERATOR LOG ..... 7-27
- EXIT ..... 7-27

EDIT MENU ..... 7-27
UNDO… ..... 7-27
CUT ..... 7-27
COPY ..... 7-28
PASTE ..... 7-28
FIND ..... 7-28
REPLACE ..... 7-28
VIEW MENU ..... 7-29
CONTROL BAR ..... 7-29
LANGUAGE ..... 7-30
UNITS ..... 7-30
ALARMS AND MESSAGES WINDOW ..... 7-31
AXES POSITIONS ..... 7-32
MODAL G CODES ..... 7-32
ALL POSITION INFORMATION ..... 7-32
RUN WINDOW GRAPHICS ..... 7-33
WINDOW ZOOMING ..... 7-33
MAINTENANCE MENU ..... 7-34
CONFIGURATION ..... 7-34
MIRROR ALIGNMENT ..... 7-44
DIAGNOSTICS ..... 7-45
RESONATOR ..... 7-48
STATISTICS ..... 7-51
LASER MENU ..... 7-52
LASER STATUS MONITOR ..... 7-52
AUTO START-UP ..... 7-54
AUTO SHUT-DOWN ..... 7-55
RESONATOR SET-UP WINDOW ..... 7-56
VARIABLES MENU ..... 7-57
LOCAL/GLOBAL VARIABLES ..... 7-57
USER I/O VARIABLES ..... 7-57
WORK OFFSETS ..... 7-58
MACHINE OFFSETS ..... 7-58
UTILITIES MENU ..... 7-59
STANDOFF CALIBRATION ..... 7-59
OPERATOR SETUP ..... 7-59
LENS CENTERING ..... 7-63
SPEED GAS ..... 7-64
WINDOW MENU ..... 7-64
CASCADE ..... 7-64
TILE ..... 7-64
ARRANGE ICONS ..... 7-64
WINDOW 1, 2 ..... 7-65
HELP MENU ..... 7-65
TOPICS ..... 7-65
MANUALS ..... 7-65
ABOUT CINCINNATI LASER SYSTEM ..... 7-66

## CUTTING PROCESS PARAMETERS

INTRODUCTION ..... 7-67
PROCESS PARAMETER WINDOW ..... 7-67
PIERCE PARAMETERS ..... 7-68
RAMPED PIERCE PARAMETERS ..... 7-69
RAPID PIERCE PARAMETERS ..... 7-70
CUT PARAMETERS ..... 7-71
DYNAMIC POWER CONTROL PARAMETERS ..... 7-72
FOCUS PARAMETERS ..... 7-74
PROCESS PARAMETER NOTES ..... 7-75
PROCESS MONITOR ..... 7-76
PIERCE-THROUGH DETECTION ..... 7-76
ADAPTIVE PIERCE MODE ..... 7-78
LOSS OF CUT DETECTION ..... 7-79

## START-UP AND SHUT-DOWN


LASER SYSTEM START-UP....7-81
CHILLER WARM-UP....7-83
SHUTTING DOWN THE LASER SYSTEM....7-83


## CALIBRATION AND ADJUSTMENT PROCEDURES


NOZZLE STANDOFF CALIBRATION....7-85
NONCONTACT STANDOFF SENSING....7-85
LASER SHOT....7-86
SETUP FOR A LASER SHOT....7-86
TAKING A LASER SHOT....7-87
CENTERING THE FOCUSING LENS....7-87
LENS CENTERING PROCEDURE....7-88
LENS FOCAL POINT LOCATION....7-91
FOCUS CALIBRATION - AUTO FOCUS HEAD....7-92
ASSIST GAS PRESSURE ADJUSTMENT....7-93
RAPID PIERCE ADJUSTMENTS....7-94


## CUTTING PROCEDURES


FIRST RUN OF PARTS....7-95
ERROR RECOVERY....7-97
CUTTING HEAD BREAKAWAY....7-97
RESTARTING A PROGRAM....7-97
BATCH MODE PROGRAM EXECUTION....7-98
MACHINE SETUP CHECKING....7-99


## AUTO FOCUS CUTTING HEAD


FOCUS CONTROL SYSTEM CONFIGURATION....7-103
AUTO FOCUS SETUP....7-104
FOCUS CONTROL SYSTEM HOME OFFSETS....7-104
FOCUS SETUP FOR LENS CENTERING....7-104
FOCUS-RELATED PROCESS PARAMETERS....7-105
AUTO FOCUS OPERATION....7-106
HOMING THE FOCUS CONTROL SYSTEM....7-106


## WINDOWS ADMINISTRATION


MANAGER ACCOUNT....7-109
LASER UTILITIES FOLDER....7-110
SOFTWARE INSTALLATION AND UPGRADES....7-110
SETUP ACCOUNT....7-111
SOFTWARE INSTALLATION AND UPGRADES....7-112
EMERGENCY REPAIR DISK....7-112
NETWORK SETUP....7-112
JOINING A DOMAIN....7-112
DRIVE DESIGNATIONS....7-112
MAPPING NETWORK DRIVES....7-113
BACKING UP THE CONTROL....7-113
MULTIPLE OPERATOR USER ACCOUNTS....7-113
TOUCHSCREEN CALIBRATION....7-113


## ALARMS AND MESSAGES


OPERATOR FYI MESSAGES....7-115
SYSTEM ALARMS....7-118
PROGRAM ERRORS....7-125
LASER ALARMS....7-128
TROUBLESHOOTING....7-129

## LASER NC PROGRAMMING


STANDARD G CODES ..... 7-131
- G00 RAPID TRAVERSE MOVE ..... 7-131
- G01 LINEAR MOVE ..... 7-132
- G02 AND G03 ARC MOVE ..... 7-132
- G04 DWELL ..... 7-134
- G09 EXACT STOP (ONE BLOCK) ..... 7-135
- G20 INCH MODE, G21 METRIC MODE ..... 7-135
- G31 POSITION CAPTURE MOVE ..... 7-135
- G40, G41, AND G42 KERF COMPENSATION ..... 7-135
- G50 AND G51 COORDINATE SYSTEM SCALING ..... 7-137
- G52 LOCAL WORK COORDINATE SYSTEM ..... 7-138
- G53 RAPID MOVE TO MACHINE COORDINATES ..... 7-138
- G54…G59 WORK COORDINATE SYSTEM SELECTION ..... 7-139
- G61 AND G64 EXACT STOP MODE ..... 7-139
- G65 SUB-PROGRAM CALL ..... 7-139
- G68 WORK COORDINATE ROTATION ..... 7-139
- G90 AND G91 ABSOLUTE AND INCREMENTAL MODE ..... 7-140
- G92 WORK COORDINATE SYSTEM SETTING ..... 7-140

CUSTOM G CODES ..... 7-141
- G84 AND G85 START CUTTING SEQUENCE ..... 7-141
- **G89** SET PROCESS PARAMETERS ..... 7-143
- G102 SET ADDITIONAL PROCESS PARAMETERS ..... 7-145
- G103 SET RAMPED PIERCE PARAMETERS ..... 7-145
- G120 AND G121 NON-STOP CUTTING ..... 7-146
- G123, G124, AND G125 VELOCITY BLENDING ..... 7-148

M CODES ..... 7-150
- M00 PROGRAM STOP ..... 7-150
- M01 OPTIONAL STOP ..... 7-150
- M02 END OF PROGRAM ..... 7-150
- M30 END OF PROGRAM / REWIND ..... 7-150
- M35 BEAM OFF ..... 7-150
- M36 Z HOLD MODE ..... 7-150
- M37 BEAM AND GAS OFF / SHUTTER CLOSE ..... 7-151
- M38 Z HOLD MODE (TIMED) ..... 7-151
- M41 Z DOWN ..... 7-151
- M42 Z UP ..... 7-151
- M43 LOWER PALLET SPECIAL FUNCTION ..... 7-151
- M44 DISABLE LOWER PALLET SPECIAL FUNCTION ..... 7-152
- M45 OPTIONAL STANDOFF MODE ..... 7-153
- M47 PARTIAL Z UP ..... 7-153
- M48 FEEDRATE OVERRIDE DISABLE ..... 7-153
- M49 FEEDRATE OVERRIDE ENABLE ..... 7-153
- M50 SWITCH PALLETS ..... 7-154
- M51 AUXILIARY OUTPUT (TIMED) ..... 7-154
- M67 OPTIONAL ASSIST GAS PRESSURE ..... 7-154
- M90 BALL TRANSFER UP ..... 7-154
- M91 BALL TRANSFER DOWN ..... 7-154
- M98 SUBPROGRAM CALL ..... 7-155
- M99 END SUBPROGRAM ..... 7-155
- M130 AND M131 Z-AXIS ANTIDIVE ..... 7-155
- M135 SPEED GAS BEAM OFF ..... 7-156
- M151 AUXILIARY OUTPUT WITH CONFIRMATION ..... 7-156

CINCINNATI MACROS ..... 7-157
- GRID MACROS ..... 7-157

CUTTING MACROS ..... 7-160
- G73 HOLE MACRO ..... 7-160
- G76 SLOT MACRO ..... 7-161
- G79 LINE MACRO ..... 7-161
- G83 OUTSIDE CIRCLE MACRO ..... 7-162
- G86 OUTSIDE RECTANGLE MACRO ..... 7-162
- G88 BOLT CIRCLE MACRO ..... 7-163
- G104 SHAPE MACRO ..... 7-163
- M2 LEAD-IN DESCRIPTION ..... 7-165
- G105 LEAD-IN MACRO ..... 7-166

PROGRAM STRUCTURE....7-167
PROGRAM NAME....7-167
PROGRAM BODY....7-167
BEAM ON AND OFF COMMANDS....7-168
PROGRAM COMMENTS....7-168
PROGRAM LINE NUMBERS....7-168
BLOCK DELETE....7-168
END OF PROGRAM....7-169
SUBPROGRAMS AND MACROS....7-169
LOCAL VARIABLES....7-169
NESTED SUB-PROGRAM CALLS....7-171
PROGRAM VARIABLES....7-171
LOCAL AND COMMON VARIABLES....7-171
SYSTEM VARIABLES....7-172
AUXILIARY FUNCTIONS....7-175
MATH FUNCTIONS....7-175
LOGIC FUNCTIONS....7-176
AUXILIARY COMMANDS....7-178
WORKPIECE EDGE DETECTION....7-180
NC CODE LIST....7-189
M CODE LIST....7-189
G CODE LIST....7-190

# CINCINNATI LASER SYSTEM SECTION 7

## CINCINNATI CL-800 SERIES LASER SYSTEM

### Supplement Manual - Section 7, Operation - for CL-800 Models

This manual applies to all CL-800 Series Laser Systems with resonators, including all frame sizes and resonator wattages.

### Control Software Versions

This document was last updated for compatibility with the following control software:

CNC/HMI software - **845601** Version **8.2**

PLC software - **845759** Version **16**

### Laser System Help Version: 1.4

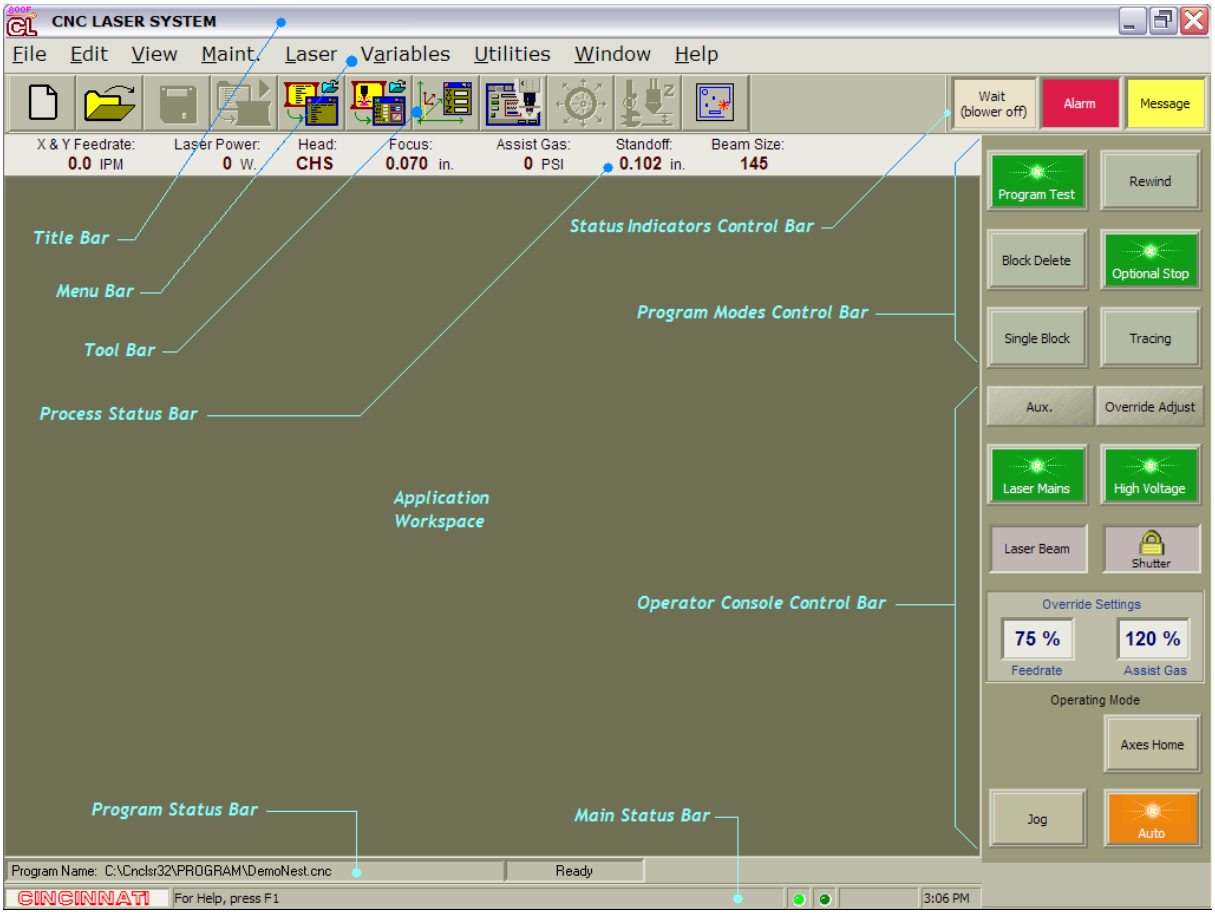
# HMI OVERVIEW

The Human Machine Interface (HMI) is the means provided for interacting with the laser system control. The HMI tools consist of the Operator Control Station front panel and side panel pushbutton controls, the LCD monitor/touchscreen, the trackball pointing device, the keyboard, the remote station, and the Laser System **CNC/HMI** software. The pushbutton controls and remote station functions are described in **SECTION 6 - MACHINE CONTROLS** of the Laser System **Operation, Safety, and Maintenance Manual**.

This chapter presents an overview of the software user interface. The touchscreen, trackball, and keyboard can be used to navigate around the user interface. The foundation of the software user interface is the *Laser System Display* window.

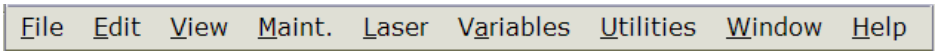
## LASER SYSTEM DISPLAY WINDOW

The *Laser System Display* window occupies the entire display screen on the operator control station. The main components of the *Laser System Display* window are the Title Bar, Menu Bar, Tool Bar, Status Indicators Control Bar, Application Workspace, Operator Console Control Bar, and Main Status Bar. A Process Status Bar, Program Modes Control Bar, and Program Status Bar may also be visible, depending on the current operating mode of the Laser System.



## MENU BAR

The Menu Bar is the row of main menu titles across the top of the *Laser System Display* window. Selecting a menu title opens a pop-up menu displaying a list of menu commands. See the **MENU COMMANDS** section for a description of each command.




## TOOL BAR

The Tool Bar, located below the Menu Bar, contains a row of buttons where each button corresponds to a menu command. The Tool Bar provides convenient, single-touch access to some of the more commonly used menu commands. The Tool Bar can be hidden or displayed by going to: **View | Control Bars.**

This list identifies which menu command is associated with each Tool Bar button:

**File | New**

**File | Open**

**File | Save**

**File | Load Current Program**

**File | Open Loaded Program**

**File | View Current Process**

**View | View Axes Positions**

**Utilities | Operator Setup**

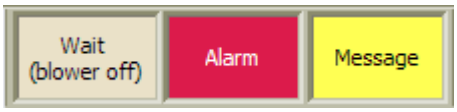
**Utilities | Lens Centering Mode**

**Utilities | Standoff Calibration**

Open the **CINCINNATI Laser Programming and Nesting** application, if installed.

## STATUS INDICATORS CONTROL BAR

The Status Indicators Control Bar, which is normally positioned to the right of the Tool Bar, consists of separate indicators for resonator state: “alarm condition present” and “operator message ready”. The left-most indicator shows the current resonator operating state. Click or touch this indicator for a shortcut to open the *Laser Status Monitor* window. The “Alarm” indicator signifies the presence of system alarms or resonator alarms when its color is red. A flashing red indicator means that one or system alarms are active. A solid red indicator means that only resonator alarms are active. A yellow “Message” indicator signifies that one or more operator FYI messages are pending. Both the “Alarm” and “Message” indicators can be used as shortcut buttons to open or close the *Alarms and Messages* window. The status indicators can be used as a shortcut to open or close the *Laser Status Monitor* window.




## PROCESS STATUS BAR

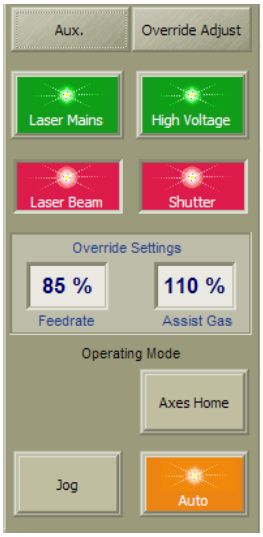
The Process Status Bar, which is located below the Tool Bar, displays critical process information such as feedrate, laser power, beam focus position, assist gas pressure, and nozzle standoff. The Process Status Bar is only displayed when the Laser System is in Auto mode. All displayed values are updated in real time. While the control is in Auto mode, the Process Status Bar can be hidden or displayed by going to **View | Control Bars | Process Status Bar**.

## APPLICATION WORKSPACE

The Application Workspace is the area (initially blank) which is below the Process Status Bar or Tool Bar and above the Program Status Bar. NC program windows, Process Parameter windows, and all other user-activated windows and dialogs will open in this area.

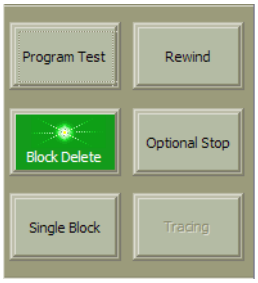
## OPERATOR CONSOLE CONTROL BAR

The Operator Console Control Bar contains the user interface controls used by the machine operator to control and monitor the basic operating mode of the Laser System. See **OPERATOR CONSOLE** in the **CONTROL BARS** section, for information on this control bar.



## PROGRAM MODES CONTROL BAR

The Program Modes Control Bar is used to manage the various program execution modes that affect how the control software executes NC programs. Like the Process Status Bar, the Program Modes Control Bar is only visible when the Laser System is in Auto mode. See **PROGRAM MODES** in the **CONTROL BARS** section for more on program modes.




## PROGRAM STATUS BAR

The Program Status Bar, located just below the Application Workspace, displays the cutting program execution status. Like the Program Modes Control Bar and the Process Status Bar, this status bar is visible only when the control is in Auto mode. The Program Status Bar is divided into two fields, the left field contains the file name of the currently loaded NC program. This field is empty when there is no NC program loaded into the control. The right field indicates the current state of the program execution subsystem. For example: “Program not loaded”, “Executing”, or “Program Stopped” may be displayed here.

## MAIN STATUS BAR

The Main Status Bar is displayed at the bottom of the *Laser Status Monitor* window, and is used to display operator help messages and other general information. The Main Status Bar is divided into several sections. The section to the right of the CINCINNATI logo displays a brief description of Menu Bar and Tool Bar button commands as these items are selected with the trackball or keyboard.

To the right of the command description is the laser control system “heartbeat” indicator. A properly functioning control system is indicated by two alternately flashing green indicators. If either indicator stops flashing or turns red, power down the system and restart it. If this does not return the indicators to their proper state, contact CINCINNATI INCORPORATED Laser Service Department.

The next field to the right is the NC Program file line number of the line where the cursor is currently located. This field is blank if an *NC Program Edit* window is not the currently active window.

In the far right section of the Main Status Bar, the current time is displayed.

# FILE TYPES

The Laser System control software gets the information it needs to execute cutting applications from two types of data files: NC Program files and Process Parameter files. A third type of file, the Batch Program file, may be used if desired, to create a list of NC Program files to be executed in the specified order.

The *File* menu commands and Tool Bar buttons, described later, can be used to open these files for viewing/editing, and to create new files. When opening an existing file or creating a new file, a window will appear in the Application Workspace, displaying the file's contents. The Title Bar of each file view window contains the name of its file.

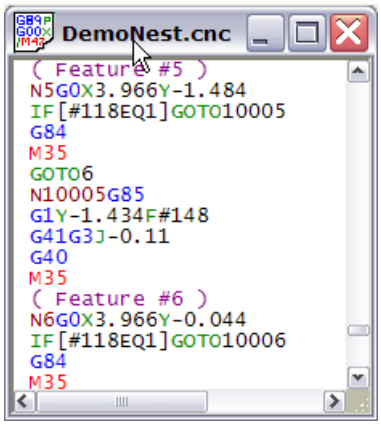
When a file is modified, it must be saved using **File | Save** or Tool Bar **Save** command before any changes will take effect. An asterisk "*" will appear next to the file name of any file that has been modified but not saved.

The following topics describe each of the three types of file and their view windows.

## NC PROGRAM FILE

The NC Program file contains the NC codes that determine the part feature geometry and control the various machine cutting functions. NC programs, which may be as simple as a single part or as complex as multiple sheets of nested parts, are typically generated by nesting/post-processing software packages such as the **CINCINNATI Laser Programming and Nesting Software** application. NC Program file names should end with ".cnc" (*ex: "Filename.cnc"*).

When an NC Program file is opened, its contents are displayed in a *Program Edit* window. *Program Edit* windows use context coloring; distinct program components are displayed with different colors (G codes are blue, M codes are red, etc).



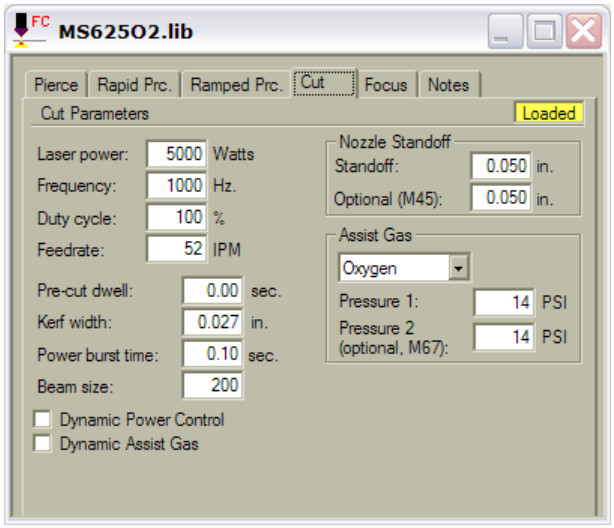

## PROCESS PARAMETER FILE

The Process Parameter file contains settings used to control the power and focusing attributes of the laser beam and the cutting assist gas. Process Parameters are stored in multiple files, with a separate file for each distinct cutting process. All CINCINNATI Laser Systems are shipped with a library of Process Parameter files containing the cutting parameter settings recommended by CINCINNATI INCORPORATED. These files are stored in the "Cnclsr32/Material" folder on the laser control's computer hard disk drive. These files can be modified to specify other cutting processes by creating new Process Parameter files with the preferred settings. Process Parameter file names should end with ".lib" (*ex: "Filename.lib"*).

**G89** blocks in the NC Program file specify which Process Parameter file(s) will be used in each cutting application. See **CUSTOM G CODES** in the **LASER NC PROGRAMMING** topic for more information on **G89**.

When a Process Parameter file is opened, the parameter settings are displayed in a *Process Parameter* window, shown on next page. See the **CUTTING PROCESS PARAMETERS** topic for more information on process parameters.

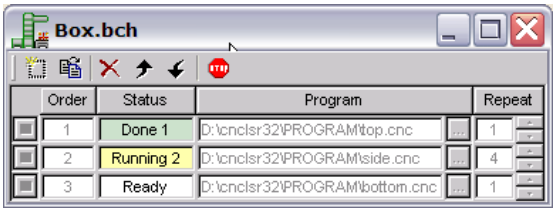
*Note: An "Active" or "Loaded" indicator:* Active Loaded *may be displayed in the upper right corner of the Process Parameter window. If a Process Parameter file that is loaded via a* **G89** *block anywhere in the active NC program is open in a window, a yellow "Loaded" indicator will be displayed. If this file represents the currently active cutting process, then an orange "Active" indicator will be displayed.*




## BATCH PROGRAM FILE

A Batch Program file is a list of NC Program files that will be loaded and executed in the order listed. It allows programs to be grouped together in a logical manner and allows the control to keep track of which programs have been completed and which program to load next. A Batch Program can contain up to one hundred program entries and each entry can be repeated up to one hundred times. Batch Program file names should end with ".bch" (*ex: "Filename.bch"*).

When a Batch Program file is opened, the parameter settings are displayed in a *Batch Program* window:




### Creating a Batch Program

To create a new Batch Program:

1. Select **File | New** and then select "**Batch**" file type in the *New Selection* window. Click "OK" to close the *New Selection* window.

2. In the new *Batch Program* window, select the "New Entry" button until the window has enough rows for all NC programs that will be added to the batch.

3. In each row, enter the NC Program file name in the **Program** column by typing it directly or using the "Browse" button to select it with the trackball.

4. Enter the number of times that each program will run in the **Repeat** column by typing it directly or by using the arrow buttons .

5. If it becomes necessary to move or delete an entire row, select the row using the "Row Select" button and then press the "Move Up" , "Move Down" , or "Delete Entry" button.

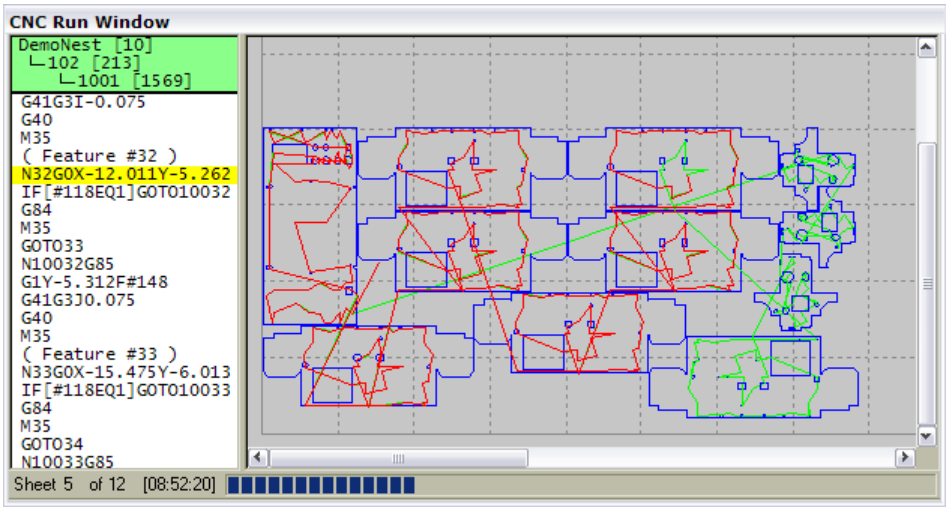
### Editing a Batch Program

Batch Program editing functions are provided in the *Batch Program* window. The editing functions are:

| | | |
|---|---|---|
| | **Insert new row** | Inserts a new NC Program file entry after the selected entry or at the end of the list. |
| | **Copy a row** | Copies the selected row and inserts the copy into the list after the selected row. |
| | **Delete Entry** | Deletes the selected entry. |
| | **Move Up** | Moves the selected entry up one row in the list. |
| | **Move Down** | Moves the selected entry down one row in the list. |
| | **Stop the Batch** | Stops a running Batch Program. |

## CNC RUN WINDOW

The *CNC Run Window* displays the execution status of the active cutting program, if a program is currently loaded. This window cannot be manually opened or closed. The control software will automatically display the *CNC Run Window* each time a part program is successfully loaded, and whenever the control enters Auto mode while a program is loaded. Likewise, the *CNC Run Window* will automatically close whenever the control enters Jog mode or Axes Home mode.




The *CNC Run Window* has two sections and a status bar. The left section displays the NC program code with the active block highlighted. To help locate the current position in an interrupted cutting program, the active subroutine name and line number are displayed in a green box at the top of the NC code section. All currently active subroutines are displayed, indented to the right to indicate the nesting depth for nested subroutines.

The right section is a graphical plot of the current sheet program. For each sheet, the status bar at the bottom of the window displays the sheet number, the remaining runtime, and a bar graph showing the percentage of completed program blocks. The plotting software interprets **M50** as the start of another sheet.

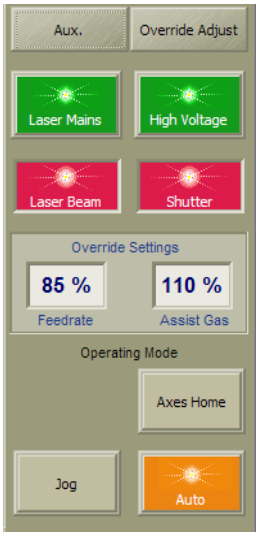
Before the program starts each sheet, the plot shows all X-axis and Y-axis rapid moves in green. Contouring moves with the laser beam on are shown in blue. Contouring moves with the laser beam off are shown in yellow. As the program runs, the plot changes the color of all completed moves to red. To change the width of the plot section, select the divider between the left and right sections and drag it to another location. To change the viewing area of the plot, use any of the various *Zoom* features. See **WINDOW ZOOMING** in the **VIEW MENU** section.

If desired, the plot section and status bar can be hidden, resulting in a much smaller *CNC Run Window*. See **RUN WINDOW GRAPHICS** in the **VIEW MENU** section.

# CONTROL BARS

## OPERATOR CONSOLE

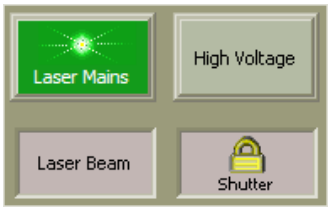
The Operator Console Control Bar contains three different groups of user controls: *Laser Resonator*, *Override Settings*, and *Operating Mode*. There are also two buttons that can be used to quickly open or close other control bars.




Selecting the "Aux." button will open the Auxiliary Functions Control Bar if it is not currently open, or close it if it is currently open. Selecting the "Override Adjust" button will open the Override Adjust Control Bar if it is not currently open, or close it if it is currently open.

### RESONATOR CONTROLS

This part of the Operator Console Control Bar contains controls related to the laser resonator.




*Laser Mains:* This control is both a button and indicator for the resonator Mains. Laser Mains will appear bright green (ON) when resonator Mains is on, and dull green or gray (OFF) when Mains is off. The "Laser Mains" button commands the main contactor of the laser resonator to open or close. The contactor supplies electrical power to the solenoids, vacuum pump, fans, and alignment laser in the resonator.

Selecting this button when Mains is off commands the contactor to close, initiating the resonator Mains startup sequence. During the startup sequence, the indicator blinks on and off; when the startup sequence is complete, the indicator will remain on. When Mains is on and laser High Voltage is off, pressing the "Laser Mains" button commands the contactor to open, turning Laser Mains off.

*Note: Laser Mains cannot be turned off when High Voltage is on. This feature prevents accidental deactivation if the touchscreen is bumped.*

*High Voltage:* The "High Voltage" button/indicator will appear bright green (ON) when resonator High Voltage is on, and dull green or gray (OFF) when High Voltage is off. If the "HIGH VOLTAGE" keyswitch on the Machine Operator Side Panel is in the LOCKED position, the indicator will be appear off, displaying an image of a padlock.

The "HIGH VOLTAGE" button commands the control to connect or disconnect electrical power to the resonator high voltage power supply. High Voltage will not activate unless Laser Mains is on, no interlock faults are present, and the "HIGH VOLTAGE" keyswitch is in the "ENABLED" position.

Selecting this button when High Voltage is off commands the control to connect power to the high voltage power supply, initiating the High Voltage startup sequence. During the startup sequence, the indicator blinks on and off; when the startup sequence is complete, the indicator will remain on. Selecting this button when High Voltage is on commands the control to remove electrical power from the resonator high voltage power supply.

*Laser Beam:* This indicator displays the On/Off status of the laser beam. The "Laser Beam" indicator is red (ON) when laser discharge current indicates that the resonator is producing a laser beam. The resonator produces a laser beam in response to requests for laser power. Laser power can be requested by the program by the **Mirror Alignment** or **Lens Centering** function or by the **Maintenance** function for Manual Power Control. When the resonator is not producing a laser beam, the indicator color is dull red or gray (OFF).

*Shutter:* This indicator displays the shutter state. When the "SHUTTER" keyswitch on the Machine Operator Side Panel is in the "ENABLED" position, the indicator color depends on the position of the shutter within the laser resonator cabinet: the "Shutter" indicator will appear red (OPEN) if the shutter is completely open, and dull red or gray (CLOSED), if the shutter is completely closed. When the "SHUTTER" keyswitch is in the "LOCK/CLOSE" position, the indicator will be appear closed, displaying an image of a padlock.

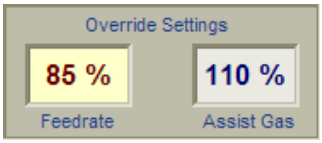
With the shutter completely closed, the laser beam is contained within the laser resonator cabinet and is not present in the beam delivery system. With the shutter open, the laser beam can be guided through the beam delivery system to the workpiece.

*Note: When the shutter is open, the Laser System control changes the color of the "Shutter" button to red and illuminates the red light on top of the gantry or the safety enclosure.*

The "Shutter" button provides manual control of the shutter only when High Voltage is off and the "SHUTTER" keyswitch is in the "ENABLED" position. Under these conditions, the shutter can be manually opened or closed by selecting the gray "Shutter" button. When the shutter is manually opened, it will remain open after the indicator is selected. To close the shutter, select the "Shutter" indicator again, or turn the "SHUTTER" keyswitch to the "LOCK/CLOSE" position.

## OVERRIDE SETTINGS

In the center section of the Operator Console Control Bar, the current values of the Jog/Feedrate and Assist Gas Override settings are displayed.




It is easy to change either override setting by selecting one of the override display boxes. This will cause the Override Adjust Control Bar to open. Once selected, that display box will change to a red text/yellow background color to indicate that the corresponding override is activated for adjustment in the Override Adjust Control Bar.

## OPERATING MODES

The bottom section of the Operator Console Control Bar contains three button/indicator controls for the primary operating mode, and an "Alternate Mode" indicator.

*Jog:* This button/indicator activates Jog mode. While the control is in Jog mode, the "Jog" indicator color is amber. In Jog mode, the axis motion controls on the Operator Control Station front panel or the Remote Station can be used to manually move ("jog") each machine axis individually. Cutting program execution is disabled while Jog mode is active. Most **Maint. | Utilities** functions, such as **Lens Centering**, are only enabled when the Laser System is in Jog mode. The control will remain in Jog mode until another mode is selected.

*Auto:* This button/indicator activates Auto mode. The Laser System must be in Auto mode in order to run cutting programs. While the control is in Auto mode, the "Auto" indicator color is amber, and most **Maint. | Utilities** functions, are disabled. The NC Program files and Process Parameter Files while the control is executing a program in Auto mode. The control will remain in Auto mode until another mode is selected.

*Axes Home:* This button/indicator activates Axes Home mode. When selected, the "Axes Home" button text changes to "Homing" and the color changes to amber. To home the axes, the main drives must be on with no system alarms present. The **Axes Homing** function begins when the operator presses the "CYCLE START" pushbutton. When all axes have moved to their reference positions, all axis motion stops, the **Axes Home** function ends and the control automatically changes to Jog mode.

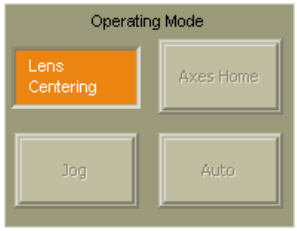
### Special Operating Mode

Special Operating Modes are alternate modes the Laser System is in while certain functions are active:

- *Lens Centering*
- *Standoff Calibration*
- *Mirror Alignment*
- *Manual Beam Control*

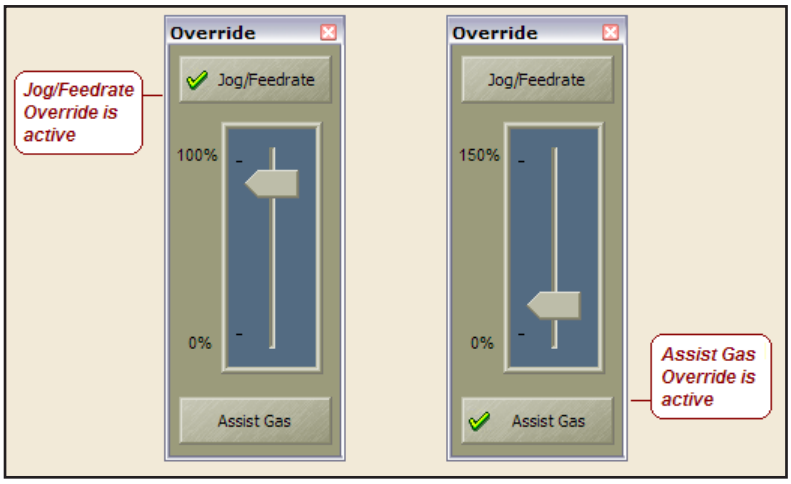
*Note: The Laser System must be in Jog mode before a "Special Operating Mode" can be activated.*

Special operating modes are typically modes in which the "CYCLE START" / "CYCLE STOP" buttons perform some unique function other than starting the execution of a cutting program. For example, Standoff Calibration is a "Special Operating Mode" since "CYCLE START" initiates the Noncontact Head standoff calibration sequence. When any of these special operating modes are active, the amber-colored "Alternate Modes" indicator will be visible, and the three primary operating mode buttons will be disabled.



## OVERRIDE ADJUST

The Override Adjust Control Bar contains a slider control can be used to adjust the setting of either the **Jog/Feedrate Override** or the **Assist Gas Override** function. Only one override setting, the active override, is adjusted at a time. The active override is selected by selecting either the "Jog/Feedrate" or the "Assist Gas" button. The currently active override is indicated by a green check mark on the corresponding button and a yellow background color for the "Override Settings" indicator in the Operator Console Control Bar.




The Override Adjust Control Bar can be opened/closed in any of the following ways:

- Select the "Override Adjust" button in the Operator Console Control Bar.
- Select either "Override Settings" indicator in the Operator Console Control Bar; this will also make the corresponding override active.
- Go to: **View | Control Bars | Overrides**

There are several ways to change the slider control setting in the Override Adjust Control Bar:

- After selecting the slider control, the "Page Up" and "Page Down" keyboard keys will change the setting in 5% increments.
- The "Up Arrow" and "Down Arrow" keyboard keys change the setting in 1% increments.
- Select the "Slider Control" indicator with the trackball pointing device and then drag it to any setting.
- Rotate the wheel on the trackball pointing device after selecting the slider control.

*Assist Gas Override:* This setting modifies the programmed assist gas pressure used for cutting. For example, if the program specifies 200 PSI and the assist gas override is set to 75%, the actual cutting pressure will be: 75% of 200 PSI = 150 PSI. This setting also affects the pressure achieved when assist gas is manually activated using one of the assist gas buttons on the Auxiliary Functions Control Bar. The assist gas override setting is adjustable from 0% to 150%.

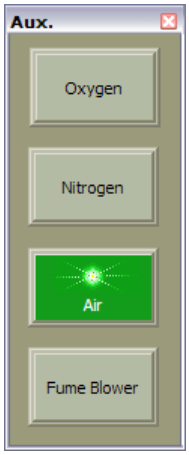
*Note: The Assist Gas Override setting does not affect the assist gas pressure used when piercing.*

*Jog/Feedrate Override:* When the NC cutting program commands the X-axis and/or Y-axis to a position with a **G01**, **G02**, or **G03** block, the actual feedrate will be the percentage of the commanded feedrate specified by the Jog/Feedrate Override setting. For example, if Jog/Feedrate Override is set to 75% and the program specifies "F400", the actual feedrate will be 75% of 400 IPM = 300 IPM. The Jog/Feedrate Override setting is adjustable from 0% to 100%. The Feedrate Override is disabled during rapid moves (**G0**) or when **M48** is active.

Jog/Feedrate Override also changes the jogging speed of the X- and Y-axes when the "RAPID TRAVERSE" button is not lit. The jogging speed at 100% is the "X/Y-axis Jog Speed" specified on the General page of the *Maintenance/ Machine Configuration* window (300 IPM or 7620 mm/min typical).

## AUXILIARY FUNCTIONS

The Auxiliary Functions Control Bar contains buttons for manually activating assist gas flow and the fume blower system. Open/close this control bar by selecting the "Aux." button on the Operator Console Control Bar, or by selecting **View | Control Bars | Aux.**




*Oxygen:* This control is both a manual activation button and indicator for oxygen assist gas. When executing a cutting program, the control automatically commands assist gas flow through the nozzle. When oxygen assist gas is flowing, the "Oxygen" indicator color changes from dull green or gray (OFF) to bright green (ON). This button is used in Jog or Auto mode to manually control the oxygen assist gas solenoid valve.

*Nitrogen:* This control is both a manual activation button and indicator for nitrogen assist gas. When nitrogen assist gas is flowing, the "Nitrogen" indicator color changes from dull green or gray (OFF) to bright green (ON). This button is used in Jog or Auto mode to manually control the nitrogen assist gas solenoid valve.

*Air:* This control is both a manual activation button and indicator for air assist gas. When air assist gas is flowing, the "Air" indicator color changes from dull green or gray (OFF) to bright green (ON). This button is used in Jog or Auto mode to manually control the air assist gas solenoid valve.

*Note: Only one assist gas can be active at a time. If nitrogen is flowing when the oxygen button is selected, the nitrogen gas solenoid will close and the oxygen gas solenoid will open, etc.*

**CAUTION**

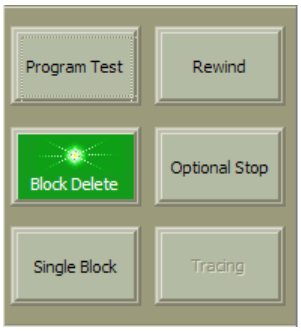
**Verify correct installation of lens drawer and nozzle tip before activating either assist gas solenoid valve. Applying assist gas pressure without proper installation of these parts can force the lens drawer or nozzle tip out of position or out of the cutting head. Unrestricted flow can also damage assist gas supply components.**

*Fume Blower:* This a manual activation button and indicator for the fume blower. Selecting this button toggles the fume blower on or off when the program is not running. The fume blower turns on automatically when the machine is cutting. When the program ends, the fume blower remains on for 60 seconds, plus the *Blower OFF delay time* specified on the Auxiliary page of the *Configuration* window, and then turns off automatically. This button is used to provide additional fume removal time. The "Fume Blower" indicator color is dull green or gray (OFF) when the fume blower is off and bright green (ON) when the fume blower is running.

## PROGRAM MODES

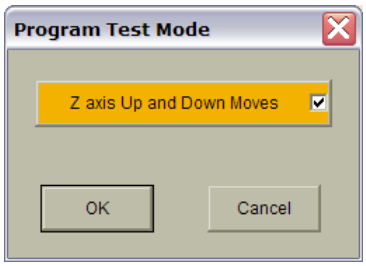
The Program Modes Control Bar contains controls that turn on or off the various program execution modes that affect how the control software executes NC programs. Each control is both a button and indicator for the corresponding mode. Like the button/indicators in the other control bars, these controls appear bright green (ON) when the corresponding mode is active, and dull green or gray (OFF) when the mode is not active.

The Laser Control Software automatically displays the Program Modes Control Bar each time the CNC control enters Auto mode. When the control enters Jog or Axes Home mode from Auto mode, this control bar will automatically close.




### PROGRAM TEST MODE

Program Test Mode will run a part program without actually cutting material. Use the "Program Test" button to activate Program Test Mode. In this mode, the laser beam and assist gas will *not* turn on during program execution. Selecting the "Program Test" button when Program Test Mode is off will cause the 'Program Test Mode' dialog box to open.




The "Z-axis Up and Down Moves" button enables or disables Z-axis motion while Program Test Mode is active. If the box is not checked, the cutting head will not move up or down, and the program will ignore the Pierce dwell. Select "OK" to close the dialog and activate Program Test Mode; select "Cancel" to close the dialog without activating Program Test Mode.

***Keyboard Shortcut:*** ***F2***

### REWIND

The "Rewind" button sets the first block of the active NC program as the next block to be executed when "CYCLE START" is pressed. Use this function to restart the program from the beginning when program execution is interrupted. The "Rewind" button is disabled while program execution is in progress. Note that this button does not stay in the green (ON) state when selected, since it does not make a new mode active.

***Keyboard Shortcut:*** ***F3***

## BLOCK DELETE MODE

While Block Delete mode is active, any block in the NC program that begins with the " / " (forward slash) character will not be executed. The "Block Delete" button activates Block Delete mode. Block Delete mode can be enabled/disabled at any time during program execution.

***Keyboard Shortcut:*** ***F4***

## OPTIONAL STOP MODE

Optional Stop mode enables the use of **M01** (or **M1**) in a program. Use the "Optional Stop" button to activate Optional Stop program execution mode. When Optional Stop mode is active, the control changes to the Cycle Stop condition when a program commands **M01**. Program execution resumes when "CYCLE START" is pressed. Optional Stop mode can be enabled/disabled at any time during program execution.

***Keyboard Shortcut:*** ***F5***

## SINGLE BLOCK MODE

In Single Block mode, only one NC Program block will be executed each time "CYCLE START" is pressed. The "Single Block" button activates Single Block program execution mode.

***Keyboard Shortcut:*** ***F6***

## TRACE MODE

The "Tracing" button activates program execution Trace mode. When Trace mode is active, the "TRACE FORWARD" and "TRACE REVERSE" buttons on the machine operator front panel are enabled. Tracing is used to recover from program interruptions; it allows the operator to step through the program in either the forward or reverse direction with the laser beam off, as long as the "TRACE FORWARD" or "TRACE REVERSE" button is pressed. See **ERROR RECOVERY** in the **CUTTING PROCEDURES** section.

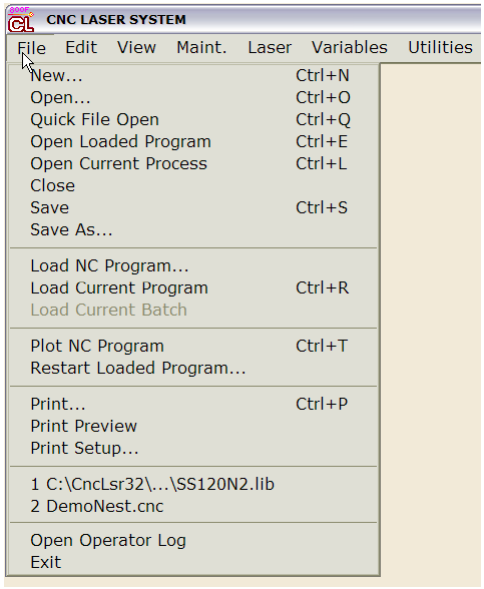
***Keyboard Shortcut:*** ***F7***

*Note: There is a limit to how far in reverse a program can be traced. As a program is running, a fixed number of executed program blocks are held in memory. This memory area, known as the "history buffer", limits how far a program can be traced in reverse. Large programs may contain more blocks than the history buffer can hold. Attempts to trace backwards beyond the last block in the history buffer will cause the alarm "End of history buffer or beginning of program reached. Reverse tracing is disabled."*

# MENU COMMANDS

## FILE MENU

In the *File* menu, various commands can be used to manage files and to activate file-related functions. Many of the *File* menu commands, such as **Open**, **Save**, and **Print**, are standard commands used by most Microsoft Windows applications, while others are for Laser System-specific functions: **Open Current Process**, **Load Current Program**, etc.




### NEW

**File | New** creates a new file and opens a new window displaying the files contents. The Laser System software works with three different types of data files (see **FILE TYPES**). Before a new file is generated, the 'New File' dialog box opens, prompting the user to specify the type of file to create. The following types of file can be created::

- NC Program File
- Process Parameter File
- Batch Program File

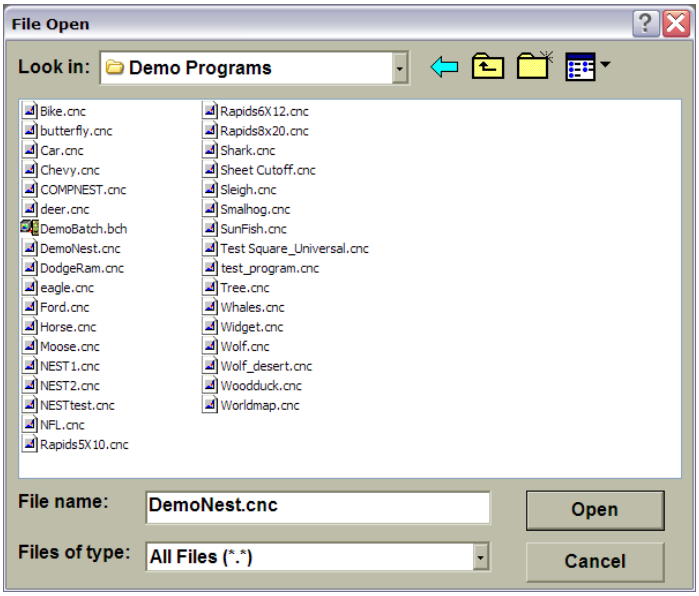
***Toolbar Shortcut:***

***Keyboard Shortcut:*** ***Ctrl + N or { Alt, F, N } sequence***

### OPEN

**File | Open** opens an existing laser file in a new window. Multiple file windows of all types can be open at the same time. This command causes the *File Open* window to appear. Use this window to select a file to open.

*Note: Use the Most Recently Used (MRU) File List to quickly reopen a file that was recently closed. See the **MRU FILE LIST** section for more information.*




Use these controls in the *File Open* window to identify the file to be opened:

*Look In:* Displays the name of the folder whose contents are displayed in the browser box below. To see how the current folder fits in the folder hierarchy, select the down arrow. To see what is inside a folder, select the folder.

*File Browser:* This is the large box below the "Look In" field that lists the folders and files in the folder specified in the "Look In" field. Use the browser to graphically select a file or a different folder. Double-clicking a folder will cause its contents to be displayed in the browser and its name to appear in the "Look In" field. The buttons above the browser window can be used to find and select folders and files:

Navigate the browser to the previously selected folder.

Navigate the browser to the folder one level higher than the currently displayed folder.

Create a new folder.

Change the browser view mode.

*File name:* Type the name of the file to be opened here or use the browser window to graphically select the file to open. Selecting a file in the browser window will cause its name to appear in the "File name" field.

*Files of type:* Lists the types of files to display in the browser. The choices are:

- All Files
- NC Program File (with *.cnc* extension)
- Process Parameter Files (with *.lib* extension)
- Batch Files (with *.bch* extension)

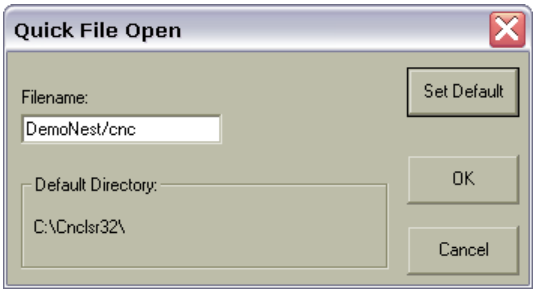
*Note: Only files with the extension listed in the "Files of Type" drop-down list are shown in the browser window.*

***Toolbar Shortcut:***

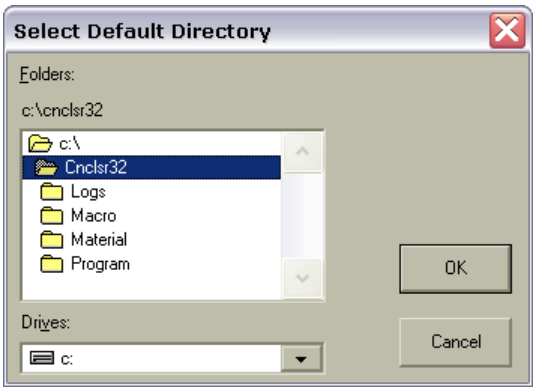
***Keyboard Shortcut:*** ***Ctrl + O or { Alt, F, O } sequence***

## QUICK FILE OPEN

**File | Quick File Open** opens existing NC Program, Process Parameter, or Batch Program files. The **Quick File Open** command will display the *Quick File Open* window. The name of the file to be opened is typed in the "Filename" field. The file must be located in the directory indicated in the 'Default Directory' box.




The "OK" button will open the file specified and close the *Quick File Open* window. The "Cancel" button will close the *Quick File Open* window without opening a file. The "Set Default" button will open the *Select Default Directory* window that will allow the user to select a new default directory. Once selected, this default directory will be saved and used until a new selection is made.




***Keyboard Shortcut:*** ***Ctrl + Q or { Alt, F, Q } sequence***

## OPEN LOADED PROGRAM

**File | Open Loaded Program** opens a window displaying the NC Program file currently loaded for execution. If no cutting program is currently loaded, this command is disabled

***Toolbar Shortcut:***

***Keyboard Shortcut:*** ***Ctrl + E***

## OPEN CURRENT PROCESS

**File | Open Current Process** opens a new window displaying the Process Parameter file currently being used for cutting. If no cutting program is currently loaded for execution, this command is disabled.

***Toolbar Shortcut:***

***Keyboard Shortcut :*** ***Ctrl + L***

## CLOSE

**File | Close** closes the active file. If the open file contains unsaved changes, the user will be prompted to save the changes before closing the file. If a file is closed that has not been named, the *Save As* window will open first, prompting the user to name the file before closing it.

*Note: If a file is closed without saving it, all changes made since the last time the file was saved will be lost.*

***Keyboard Shortcut:*** ***Ctrl + F4 or { Alt, F, C } sequence***

***"X" button on the window's caption bar :***

## SAVE

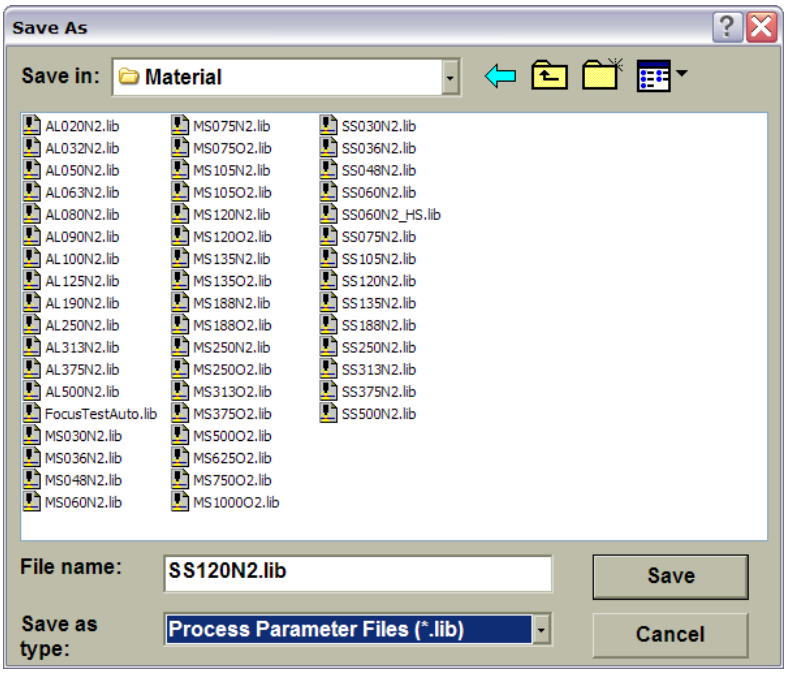
**File | Save** saves any changes to the active file. The contents of the file will be written to its current location with its current file name. When a new file is saved for the first time, the *Save As* window will open first, prompting the user to name the file.

***Toolbar Shortcut:***

***Keyboard Shortcut:*** ***Ctrl + S or { Alt, F, S } sequence***

## SAVE AS

**File | Save As** saves a new file with the specified name, or saves the contents of the active file to a different name and/or location. This command causes the *Save As* window to open. Use this window to specify the file name and location.



Use these controls in the 'Save As' dialog box to specify the name of the file and its location:

*Save in:* Displays the name of the folder whose contents are displayed in the browser box below it. To see how the current folder fits in the folder hierarchy, select the down arrow. To see what is inside a folder, select the folder.

*File Browser:* This is the large box below the "Save in" field that lists the folders and files in the folder specified in the "Save In" field. Use the browser to graphically select a file or a different folder. Double-clicking a folder will cause its contents to be displayed in the browser and its name to appear in the "Save in" field. The buttons above the browser window can be used to find and select folders and files:

Navigate the browser to the previously selected folder.

Navigate the browser to the folder one level higher than the currently displayed folder.

Create a new folder.

Change the browser view mode.

*File name:* Enter the file name here or use the browser window to graphically select an existing file name. Selecting a file in the browser window will cause its name to appear in the "File name" field. If the file name entered here has no extension, the extension listed in the "Save as type" drop-down list will be automatically added to the file name when it is saved.

*Save as type:* Specifies the type of file being saved. The list automatically includes the file type corresponding to the document in the active window as the default type. For example, if the active window is a Process Parameter file, the box will list "Process Parameter Files (*.lib)" as the file type.

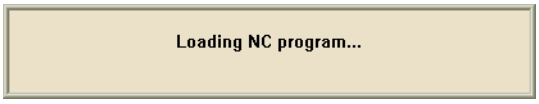
*Note: Only files with the extension listed in the "Files of Type" drop-down list are shown in the browser window.*

***Keyboard Shortcut:*** ***{ Alt, F, A } sequence***

## LOAD NC PROGRAM

**File | Load NC Program** allows the user to specify an NC Program file to load into program execution memory. Each program must be loaded into memory before it can be run. Once a program is loaded, it can be run multiple times without being loaded again. Note that only one program can be loaded at a time. This command opens the *Load NC Program to Execute* window, from which box to select a file to load. The *Load NC Program to Execute* window is similar to the window used to open a file. See the **OPEN** topic in the **FILE** section for help with using this window.

Some very large cutting programs may require several seconds to load. The following message window will be displayed while the control is busy loading a program:



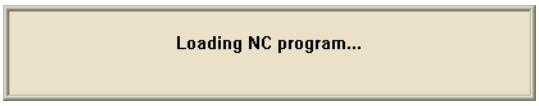

*Note: The Laser System user interface is disabled until the control finishes loading the program.*

***Keyboard Shortcut:*** ***{ Alt, F, L } sequence***

## LOAD CURRENT PROGRAM

**File | Load Current Program** loads the NC Program in the currently active *Program Edit* window into program execution memory. This command is enabled only when an NC Program file is open in the currently active window. Each program must be loaded into memory before it can be run. Once a program is loaded, it can be run multiple times without being loaded again. Note that only one program can be loaded at a time.

Some very large cutting programs may require several seconds to load. The following message window will be displayed while the control is busy loading a program:




*Note: The Laser System user interface is disabled until the control finishes loading the program.*

***Toolbar Shortcut:***

***Keyboard Shortcut:*** ***Ctrl + R***

## LOAD CURRENT BATCH

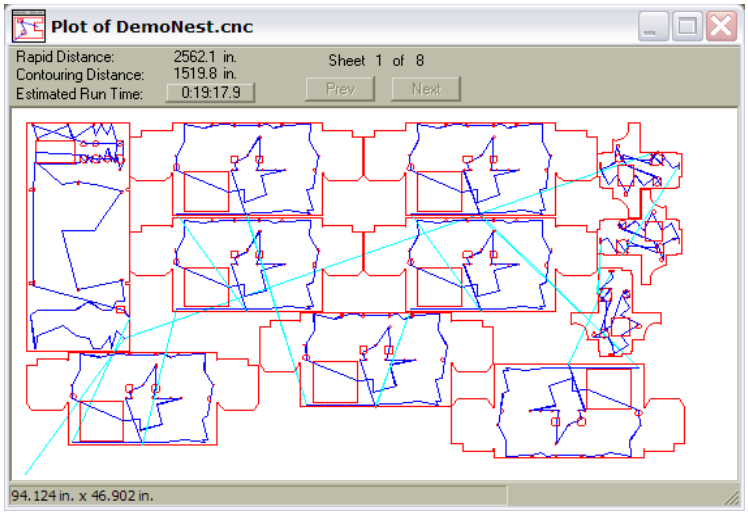
**File | Load Current Batch** loads the currently open Batch Program file for execution. This command is enabled only when a Batch Program file is open in the currently active window. See the **BATCH PROGRAM FILE** topic in the **FILES TYPES** section for more about Batch Program files.

## PLOT NC PROGRAMS

When the active window is an *NC Program Edit* window, the **File | Plot NC Program** command can be used to display a graphical plot of the cutting program, one sheet at a time.

If the program has no syntax errors, this command will open a new *Program Plot* window showing the programmed tool path. The plot represents where the cutting head would move if the operator ran the program. If the program has a syntax error, selecting "Plot NC Program" will only display the error message, and the cursor in the *Program Edit* window will be positioned on the line containing the error.

The *Program Plot* window uses different colors to display two types of rapid traverse moves (light blue for Z-axis up and dark blue for Z-axis down), and two types of contouring moves (red for laser on and green for laser off). When the program uses more than one sheet, the "Next" and "Prev" buttons change the plot window to display another sheet. The plotting software interprets **M50** in the program as the beginning of another sheet.

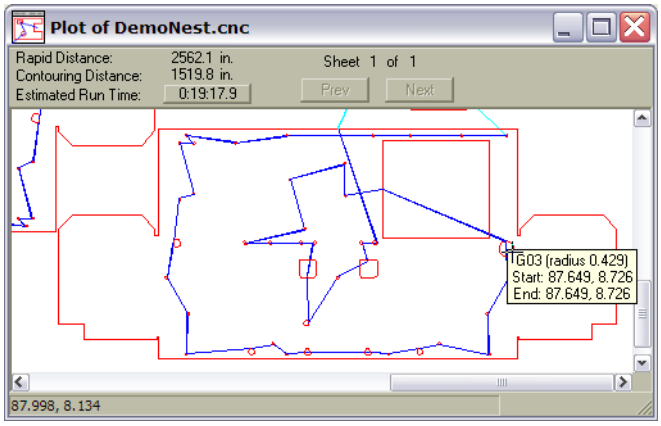



The top section of the *Program Plot* window displays the total *Rapid Distance, Contouring Distance*, and *Estimated Run Time* for each sheet. The estimated run time does not include time to exchange pallets (**M50**) or the effect of the Feedrate Override setting. The plotting function assumes that the pallets are in position and the machine is ready to run.

At the bottom of the *Program Plot* window, a status bar displays the overall X-axis and Y-axis dimensions of the cutting program.

To change the plot window magnification, use the **Zoom** functions (see **WINDOW ZOOMING** in the **VIEW MENU** section).

The *Program Plot* window can also display information about individual lines and arcs in a program. When the cursor is positioned on a line or arc, a small pop-up window displays the command type (**G00**, **G01**, **G02**, or **G03**) with the X and Y machine coordinates of the start and end. The pop-up window also displays the radius for a **G02** or **G03** arc.



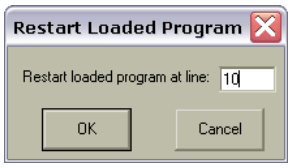

***Keyboard Shortcut:*** ***Ctrl + T***

## RESTART LOADED PROGRAM

**File | Restart Loaded Program** activates the **Program Restart** function, which allows a program to be restarted at a specified line number. This function is intended for use when long programs are terminated before completion and the normal program recovery method cannot be used. Program Restart is particularly useful with the **Automatic Laser Shut-down** function (see **Auto SHUT-DOWN**, in the **LASER MENU** section), which will display the last absolute line number executed when a program has not run to completion.

*Note: Restart Loaded Program will be enabled only if a program is loaded, the machine is in Auto mode, and is not currently running a program. At all other times, this menu command will appear grayed, indicating the **Program Restart** function is disabled.*

When this command is selected, the *Restart Loaded Program* window will be displayed, prompting the user for an absolute line number to restart the program at.




*Note: Absolute line numbers identify the sequential position of a row or block of code in a program file. This number is always relative to the first line in the program, and is not necessarily the same as the optional program line number, "Nxxxxx".*

The absolute line number of a part program is displayed in the Main Status Bar when the program file is open in a *Program Edit* window, and that window is the active window. To determine the absolute line number corresponding to a given block in a program file, open the file, move the cursor to the desired block and observe the line number in the status bar.

*Note: If any window other than a **Program Edit** window is the active window, the line number will not be displayed in the Main Status Bar.*

Selecting "Cancel" will close the dialog and abort the restart operation. Selecting "OK" will cause the control to search through the program from the beginning until it finds the desired line number. A message window will be displayed, stating: "Please wait while restart line number is found". This may take a few seconds to finish. If the line number is not reached before an **M30**, **M02**, or **M99** in a main program, an error will be displayed stating "Restart End" and the program will be rewound.

After the line number is reached, the *CNC Run Window* will be updated to show the next line to be executed. Press "CYCLE START" and the Z-axis will move to the full up position, then the X- and Y-axes will move from the current position to the beginning of the current block at 500 IPM. Normal program execution will resume from there. If the program would normally have been cutting at this point, the Z-axis will move down and cutting will resume, otherwise the head will remain up until the program commands it down to start cutting.

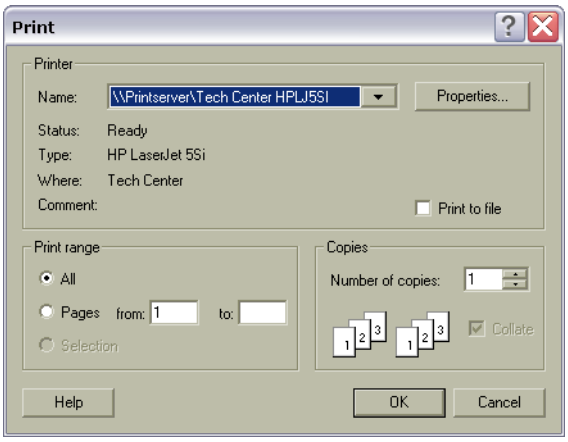
The following restrictions apply to the **Program Restart** function:

- The **Program Restart** function works best with "straight line" programs, for example, programs that do not contain subroutine calls, macro calls, or "while" loops.
- A program cannot be restarted inside a macro.
- A program can be restarted inside a subroutine as long as that subroutine is located in the same file as the main program. Note, however, if this is done, the program will restart at the first instance the subroutine is called.
- If a program is restarted inside a "while" loop, the program will restart at the first iteration through the loop.
- If the work coordinate system of a program depends on the starting position of the cutting head (for example, a program beginning with **G92 X0 Y0**), then the program will not restart at the correct work coordinates unless the cutting head has not moved since the program was terminated.

***Keyboard Shortcut:*** ***{ Alt, F, E } sequence***

## PRINT

**File | Print** uses the specified printer to print the document displayed in the active window. This command opens the *Print* window containing the following controls:




*Note: The **File | Print** command is enabled only when the currently active window contains a document type that supports printing, such as program and Process Parameter documents.*

Use these controls in the *Print* window to specify how the file is to be printed:

*Name:* This box displays a list of printers connected to the computer.

*Properties:* Select this button to set up options for the selected printer. The options available depend on the features of the printer specified in the "Name" field.

*Print to file:* Check this box to print the document to a file instead of routing it directly to a printer. It will prompt the user to specify the file name and location.

*Print range:* This section specifies what portion of the document to print: the entire document, specific pages, or the selection highlighted.

*Number of copies:* This box specifies how many copies of the document to print.

*Collate:* When printing more than one copy, this selection specifies whether the copies will be collated.

***Keyboard Shortcut:*** ***Ctrl + P or { Alt, F, P } sequence***

## PRINT PREVIEW

Use **File | Print Preview** to display the document in the active window, as it will appear when printed.

*Note: This command is enabled only when the currently active window contains a document type that supports printing, such as NC Program and Process Parameter documents.*

The **Print Preview** command opens a *Print Preview* window in which one or two pages of the active document are displayed in their printed format. A toolbar at the top of the *Print Preview* window contains buttons for changing the page and scaling of the preview:

*Print:* Open the 'Print' dialog box to start a print job.

*Next Page:* Preview the next printed page.

*Prev. Page:* Preview the previous printed page.

*One Page/Two Page:* Toggle between previewing one or two pages at a time.

*Zoom In:* Take a closer look at the printed page.

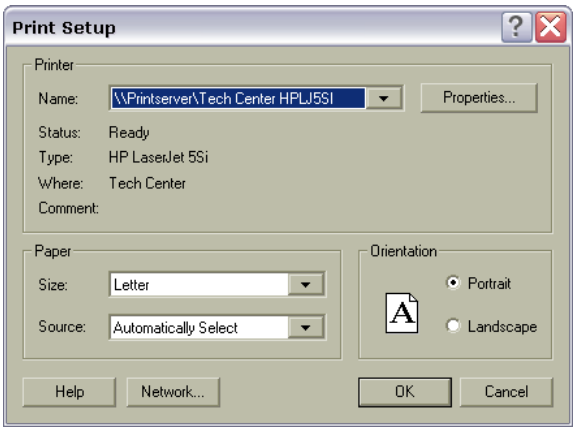
*Zoom Out:* Take a larger look at the printed page.

*Close:* Return from Print Preview to the editing window.

***Keyboard Shortcut:*** ***{ Alt, F, V } sequence***

## PRINT SETUP

This **File | Print Setup** command specifies where and how to print a document. The **Print Setup** command opens the *Print Setup* window. Use the controls in this dialog box to select a printer and printing options.




*Name:* This field displays a list of printers connected to the computer.

*Properties:* Select this button to set up options for the selected printer. The options available depend on the features of the printer specified in the "Name" field.

*Paper Size:* Specifies the size of paper on which to print the document.

*Paper Source:* Specifies where the paper is located in the printer. Different printer models support different paper sources; such as the upper tray, envelope feed and manual feed.

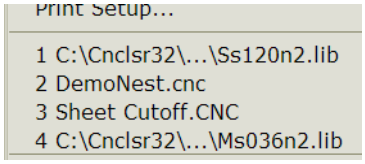
*Orientation:* Specifies whether the document should be printed with its top along the short edge of the paper (portrait) or along the long edge of the paper (landscape).

*Network:* Select this button to connect to a network shared printer.

***Keyboard Shortcut:*** ***{ Alt, F, R } sequence***

## MRU FILE LIST

When closing a Process Parameter file, NC Program file or Batch Program file, that file name is added to the Most Recently Used (MRU) File List. This list, which is displayed near the bottom of the *File* menu, can contain up to six file names. To quickly reopen a file that was recently closed, simply select the corresponding item from the MRU File List.




***Keyboard Shortcut:*** ***{ Alt, F, 1 } , { Alt, F, 2 } , { Alt, F, 3 } or { Alt, F, 4 } sequence***

### OPEN OPERATOR LOG

The Operator Log File is a text file that the user can use as a simple note pad. It can be quickly opened by using the **File | Open Operator Log** command.

### EXIT

**File | Exit** will terminate the CNC/HMI program. Before exiting, always make sure the servo drives are off, resonator High Voltage is off and any open file changes have been saved. The **Exit** command is disabled while cutting program execution is in progress.

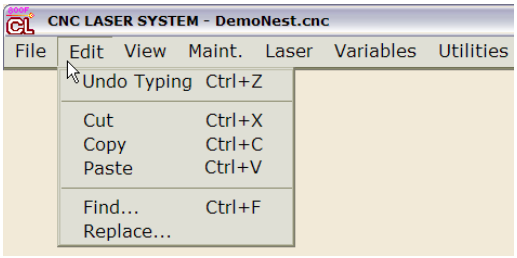
***Keyboard Shortcut:*** ***Alt + F4 or { Alt, F, X } sequence***

***"X" button on Laser Display window Title Bar:***

***Double-click icon on Laser Display window Title Bar:***


## EDIT MENU

Use the *Edit* menu commands for common text editing functions. These commands are only enabled when an *NC Program* window is the currently active window.




### UNDO…

**Edit | Undo…** will reverse the last editing action, if possible. The name of the command changes depending on the last action (for example, "Undo Typing", "Undo Cut", etc.).

***Keyboard Shortcut:*** ***Ctrl + Z or Alt + Backspace or { Alt, E, U } sequence***

### CUT

**Edit | Cut** will remove the currently selected text from the file and put it on the clipboard. Text in the clipboard can be pasted to another location. This command is unavailable if no text is currently selected. Cutting text to the clipboard replaces anything that may have been previously stored on the clipboard.

***Keyboard Shortcut:*** ***Ctrl + X or { Alt, E, T } sequence***

## COPY

**Edit | Copy** will place a copy of the selected text on the clipboard without removing the selection from the active file. This command is unavailable if no data is currently selected. Copying text to the clipboard replaces any previous clipboard contents.

***Keyboard Shortcut:*** ***Ctrl + C or { Alt, E, C } sequence***

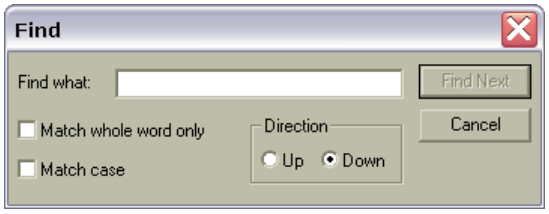
## PASTE

**Edit | Paste** inserts a copy of the clipboard contents at the insertion point in the active file window. This command is unavailable if the clipboard is empty.

***Keyboard Shortcut:*** ***Ctrl + V or { Alt, E, P } sequence***

## FIND

**Edit | Find** opens the *Find* window, allowing the user to specify text to search for in the active file. The *Find* window contains check boxes to specify matching the case of the searched letters (uppercase or lowercase), and whole words only. Use the direction buttons to specify searching "Down" (towards end of file) or "Up" (towards beginning of file). After entering text to search for in the "Find what" field, click "Find Next" to find the first occurrence of the specified text.

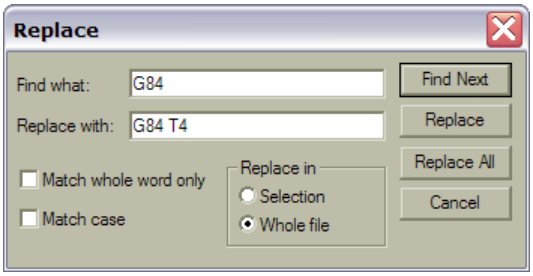



*Note: After "Find Next" has been selected, the* ***Find*** *window will close. The* ***F3*** *key on the keyboard can now be used as a "Find Next" shortcut, finding additional instances of the specified text.*

***Keyboard Shortcut:*** ***Ctrl + F or { Alt, E, F } sequence***

## REPLACE

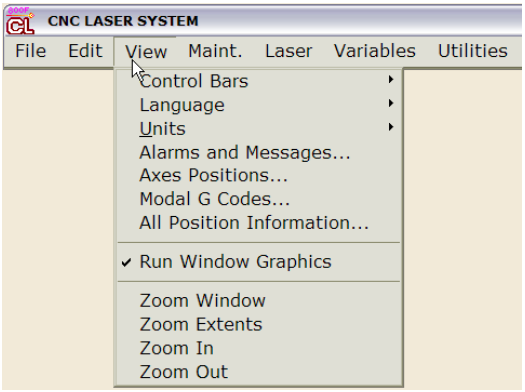
**Edit | Replace** will open the *Replace* window, which functions similar to the *Find* window, except that when specified text is found, it can be replaced with another text string specified in the "Replace with" field. Use the "Replace" button to replace only the text selection that is currently highlighted. Use "Replace All" to replace all occurrences of the specified text in the entire file at once.




***Keyboard Shortcut:*** ***{ Alt, E, R } sequence***

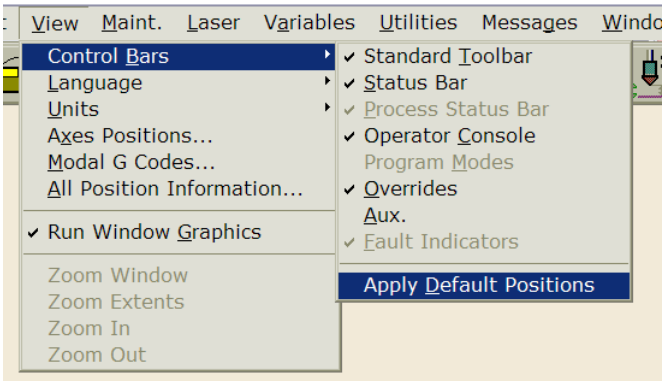
## VIEW MENU

The *View* menu allows the user to change how the user interface is displayed and configured, and open various status display windows.




### CONTROL BAR

**View | Control Bars** displays a fly-out menu with selections that display or hide the various control bars. A control bar is a group of user interface buttons and/or indicators that can be displayed or hidden with one command. A check mark appears next to the menu item for each control bar that is currently displayed.




*Note: At times, changes to the state of a control bar will not be permitted, in which case that menu item will be disabled. For example, the Process Status Control Bar cannot be displayed when the control is in Jog mode.*

*Standard Toolbar:* The Standard Toolbar contains buttons for common commands, such as "File Open" and "Load Current Program". See **TOOL BAR** in the **HMI OVERVIEW** section for more details.

*Status Bar:* The main status bar is at the bottom of the application window. It contains operator help messages, the program line number, and the time. See **MAIN STATUS BAR** in the **HMI OVERVIEW** section for more details.

*Process Status Bar:* The Process Status Bar displays actual cutting parameters. See **PROCESS STATUS BAR** in the **HMI OVERVIEW** section for more details.

*Operator Console:* This control bar contains controls for operating mode, override settings and resonator control. See **OPERATOR CONSOLE** in the **CONTROL BARS** section for more details.

*Program Modes:* The Program Modes Control Bar contains button/indicators for the various program execution modes. See **PROGRAM MODES** in the **CONTROL BARS** section for more details.

*Overrides:* This submenu item displays or hides the Override Adjust Control Bar, used to change the Assist Gas pressure or Feedrate override setting. See **OVERRIDE ADJUST** in the **CONTROL BARS** section for more details.

*Aux.:* This submenu item displays or hides the Auxiliary Functions Control Bar, which contains "Assist Gas" and "Fume Blower" buttons/indicators. See **AUXILIARY FUNCTIONS** in the **CONTROL BARS** section for more details.

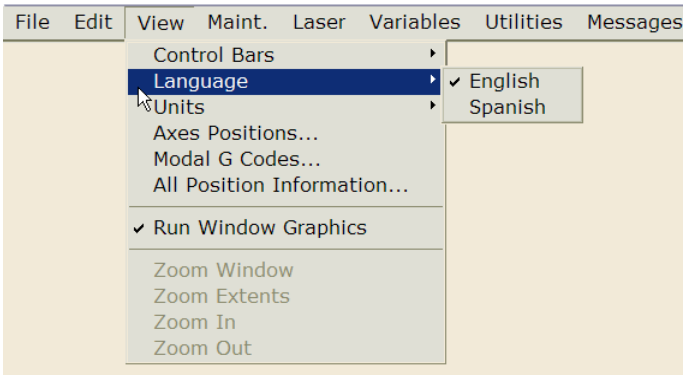
*Fault Indicators:* The three "Fault" indicators are displayed as one control bar. If the control bar is moved and then closed, this command will open the control bar at the default screen location (on the right side of the tool bar). See **STATUS INDICATORS TOOL BAR** in the **HMI OVERVIEW** section for more details.

*Apply Default Locations:* This submenu command displays all control bars at their default screen locations. Use this command to restore any control bars that were moved or closed.

***Keyboard Shortcut:*** ***{ Alt, V, B, D } sequence***

## LANGUAGE

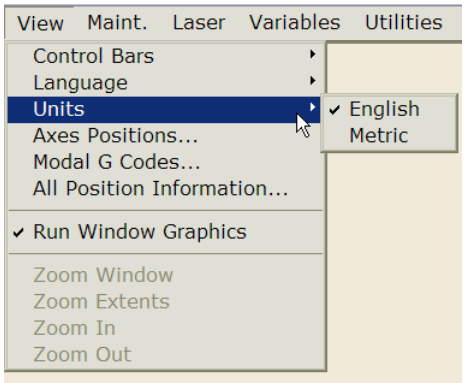
**View | Language** opens a fly-out menu showing the available user interface languages. Use this command to change the language used for all user interface text display by the Laser System **CNC/HMI** application. A check mark will appear next to the currently active language. Before selecting a new language, save/close any open files. The **CNC/HMI** application will automatically close after the new language is selected. When the **CNC/HMI** application is restarted, the new user interface language will be active.




***Keyboard Shortcut:*** ***{ Alt, V, L } sequence***

## UNITS

**View | Units** opens a fly-out menu to configure the user interface for displaying English or Metric units.



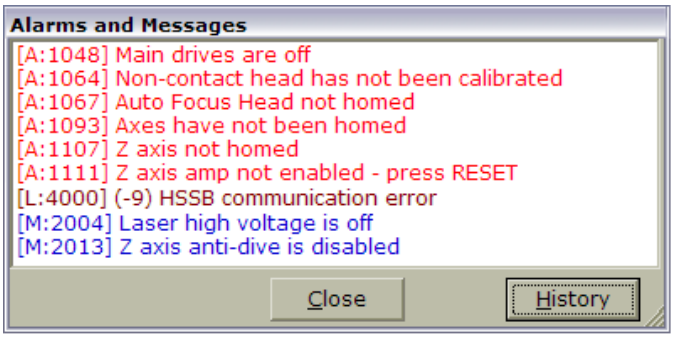

When this command is selected, the *English/Metric* menu appears with a check mark beside the currently selected display units. This selection does NOT affect program execution in inch mode (**G20**) or metric mode (**G21**).

***Keyboard Shortcut:{ Alt, V, U } sequence***

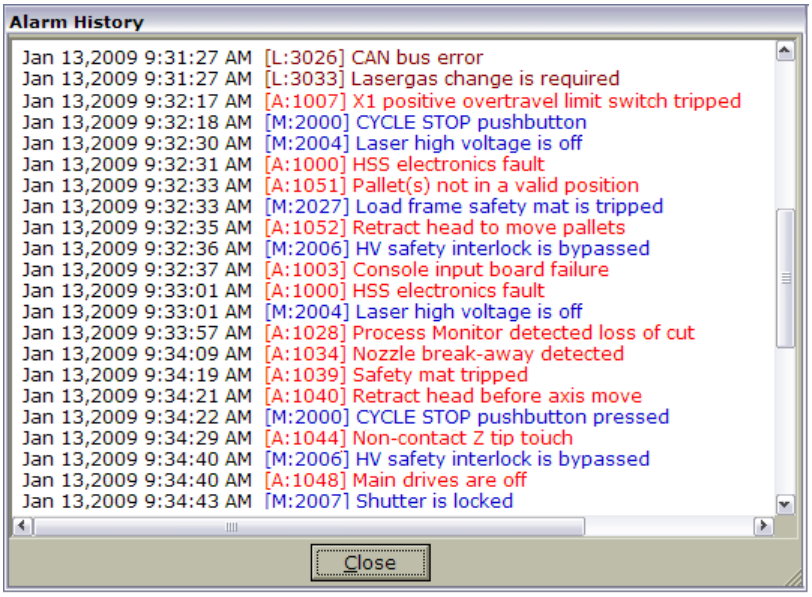
## ALARMS AND MESSAGES WINDOW

The *Alarms and Messages* window displays a list of all currently active alarm and FYI messages. Messages are displayed in different colors according to the type of message. To further assist in identifying each message condition, a [letter: number] code precedes the text for each message.

- System alarm messages are displayed in Red, and preceded by a unique [A:nnnn] identifier.
- FYI messages are displayed in Blue, and preceded by a unique [M:nnnn] identifier.
- Resonator alarm messages are displayed in Maroon, and preceded by a unique [L:nnnn] identifier.


Alarms and Messages
[A:1048] Main drives are off
[A:1064] Non-contact head has not been calibrated
[A:1067] Auto Focus Head not homed
[A:1093] Axes have not been homed
[A:1107] Z axis not homed
[A:1111] Z axis amp not enabled - press RESET
[L:4000] (-9) HSSB communication error
[M:2004] Laser high voltage is off
[M:2013] Z axis anti-dive is disabled
Close
History

The "History" button in the *Alarms and Messages* window can be used to open an *Alarm History* window that displays a list of the 250 most recent alarm and operator messages.


Alarm History
Jan 13,2009 9:31:27 AM [L:3026] CAN bus error
Jan 13,2009 9:31:27 AM [L:3033] Lasergas change is required
Jan 13,2009 9:32:17 AM [A:1007] X1 positive overtravel limit switch tripped
Jan 13,2009 9:32:18 AM [M:2000] CYCLE STOP pushbutton
Jan 13,2009 9:32:30 AM [M:2004] Laser high voltage is off
Jan 13,2009 9:32:31 AM [A:1000] HSS electronics fault
Jan 13,2009 9:32:33 AM [A:1051] Pallet(s) not in a valid position
Jan 13,2009 9:32:33 AM [M:2027] Load frame safety mat is tripped
Jan 13,2009 9:32:35 AM [A:1052] Retract head to move pallets
Jan 13,2009 9:32:36 AM [M:2006] HV safety interlock is bypassed
Jan 13,2009 9:32:37 AM [A:1003] Console input board failure
Jan 13,2009 9:33:01 AM [A:1000] HSS electronics fault
Jan 13,2009 9:33:01 AM [M:2004] Laser high voltage is off
Jan 13,2009 9:33:57 AM [A:1028] Process Monitor detected loss of cut
Jan 13,2009 9:34:09 AM [A:1034] Nozzle break-away detected
Jan 13,2009 9:34:19 AM [A:1039] Safety mat tripped
Jan 13,2009 9:34:21 AM [A:1040] Retract head before axis move
Jan 13,2009 9:34:22 AM [M:2000] CYCLE STOP pushbutton pressed
Jan 13,2009 9:34:29 AM [A:1044] Non-contact Z tip touch
Jan 13,2009 9:34:40 AM [M:2006] HV safety interlock is bypassed
Jan 13,2009 9:34:40 AM [A:1048] Main drives are off
Jan 13,2009 9:34:43 AM [M:2007] Shutter is locked
Close

Any system alarm or laser (resonator) alarm causes machine operation to stop. Conditions that cause alarms must be corrected before operation can continue. When an alarm condition exists, the "Alarm" indicator on the Status Indicators Control Bar will be colored red.

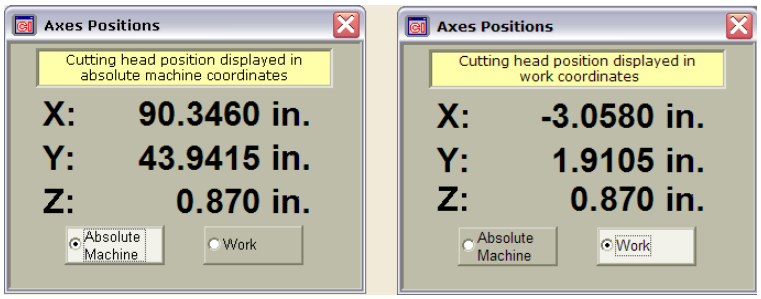
Conditions that cause FYI messages do not prevent machine operation but are useful to the operator as a reminder of some required action or that the machine is in a special mode (Example: Remote Station enabled). When an FYI message is pending, the "Message" indicator on the Status Indicators Control Bar will be colored yellow.

***Status Indicators Control Bar :*** Alarm ***or*** Message

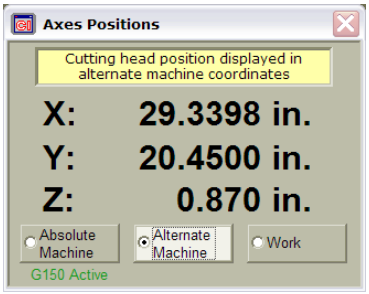
***Keyboard Shortcut: { Alt, V, A} sequence***

## AXES POSITIONS

**View | Axes Positions** opens the *Axes Positions* window, which is used to monitor the current position of the cutting head in Machine or Work coordinates. The displayed positions change when any axis moves. Select either the "Absolute Machine" or "Work" button to specify the type of displayed coordinates. The display field above the coordinates also describes the selection.




When the *Machine Offsets* window specifies non-zero Offset Distances for the alternate machine coordinate system, the *Axes Positions* window has another button to select those coordinates:



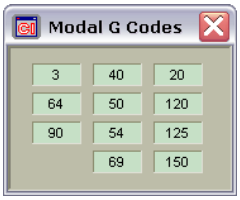

When the *Axes Positions* window displays a button for the **Alternate Machine Coordinate System**, the window also indicates the **G** code of the active machine coordinate system at the bottom (**G150** standard or **G153** alternate).

***Toolbar Shortcut:***

***Keyboard Shortcut:*** ***{ Alt, V, X } sequence***

## MODAL G CODES

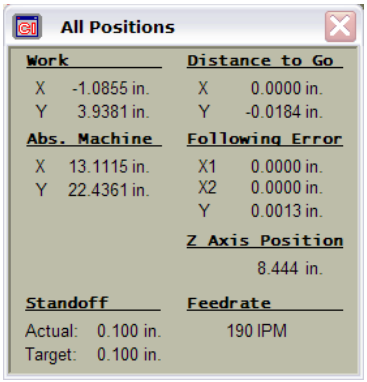
**View | Modal G Codes** opens the *Modal G Codes* window. This window displays the active **G** code from each modal group while a program is running. The software updates the window every 200 milliseconds.




***Keyboard Shortcut:*** ***{ Alt, V, M } sequence***

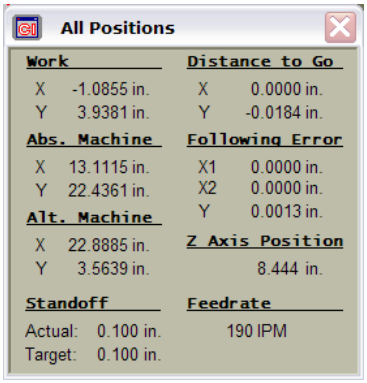
## ALL POSITION INFORMATION

**View | All Position Information** opens the *All Positions* window, which displays the position of the cutting nozzle in Machine and Work coordinates at the same time. The displayed positions change when any axis moves. This window also displays the "Distance to Go" and "Following Error" of the current motion command, the combined X-axis and Y-axis Feedrate, and the target and actual nozzle Standoff.




*Note: When the Z-axis raises the nozzle greater than 0.040 inches (1 mm) above the target standoff position, the actual standoff display reads "–".*

When the *Machine Offsets* window specifies non-zero Offset Distances for the alternate machine coordinate system, the *All Positions* window includes the cutting head position in the alternate machine coordinate system.




***Keyboard Shortcut:*** ***{ Alt, V, P } sequence***

## RUN WINDOW GRAPHICS

The **View | Run Window Graphics** command enables or disables the graphic display of the loaded sheet program plot in the CNC Run Window. When the function is active, a check mark will be displayed next to "Run Window Graphics" in the *View* menu, and the *CNC Run Window* will display the plot graphics.

***Keyboard Shortcut:*** ***{ Alt, V, G } sequence***

## WINDOW ZOOMING

The **Zoom** commands at the bottom of the *View* menu can be used to change the displayed area of an *NC Program Plot* window or the *Run Window Graphics* plot.

*Zoom Window*: Use this command to select a rectangular area to magnify. The **Zoom Window** command changes the cursor appearance to that of the 'Zoom Area' selection cursor: Drag this cursor to draw a rectangular outline around the area of interest. After the mouse button has been released to complete the drag, the outlined rectangle will fill the plot window.

*Zoom Extents*: Used in the *CNC Run Window* to scale the plot area to match the machine pallet size. In a *Program Plot* window, this command will scale the display so that the cutting program completely fills the plot window. This is the default display mode for both the *CNC Run Window* and *Program Plot* window.

*Zoom Parts:* Used in the *CNC Run Window* to scale the plot area so that it is completely filled by the cutting program parts.

*Zoom In:* Zooming-in magnifies the center portion of the plot by one zoom increment.

*Zoom Out:* Zooming-out displays a larger area than the previous plot. This makes the displayed program plot one zoom increment smaller.

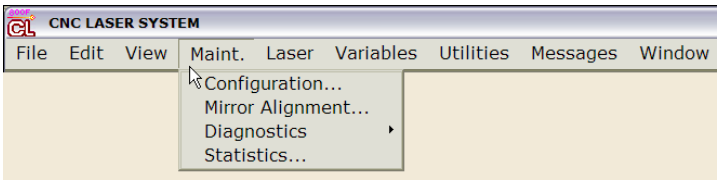
| | | |
|---|---|---|
| ***Keyboard Shortcut:*** | Ctrl + Home = | Zoom Extents |
| | Ctrl + Page Up = | Zoom In |
| | Ctrl + Page Down = | Zoom Out |

***Pointing Device :*** Clicking the right button in a plot window will display a pop-up menu containing the Zoom commands.

***Pointing Device :*** When the cursor is positioned in a plot window, rotating the mouse wheel will activate the **Zoom-in** or **Zoom-out** function.

## MAINTENANCE MENU

Use the *Maintenance (Maint.)* menu for access to various machine maintenance and diagnostic functions.




### CONFIGURATION

**Maint. | Configuration** opens the *Machine Configuration* window, which is used to view and modify system parameters and installed option settings. The *Machine Configuration* window is actually several dialogs, or pages, housed in a common frame window. Along the top of the frame window is a row of tabs, one for each configuration page. Select any tab to open the corresponding page and close the previously open page.

*Note: The **Machine Configuration** command is disabled while the laser control is in Auto mode.*

The *Machine Configuration* window is divided into these sections; one window page covers each section:

- General Configuration
- Option Configuration
- X Axis Configuration
- Y Axis Configuration
- Z-axis Configuration
- Auxiliary Configuration
- I/O Labels Configuration
- Serial Port Configuration

*Note: The 'Machine Configuration' dialog is for use primarily by CINCINNATI Service personnel to modify system parameters not normally changed by the operator. Changing configuration parameters by non-CINCINNATI personnel is restricted to the parameters specified in the "Edit" button function description.*

## Button Functions

*OK:* Closes *Machine Configuration* window. Any changes will be saved.

*Cancel:* Closes *Machine Configuration* window without saving changes.

*Edit:* When the *Machine Configuration* window is first opened, all parameter editing is disabled. Selecting this button will enable editing if the edit access requirements listed are met.

**Administrator**: All values on all pages can be edited.

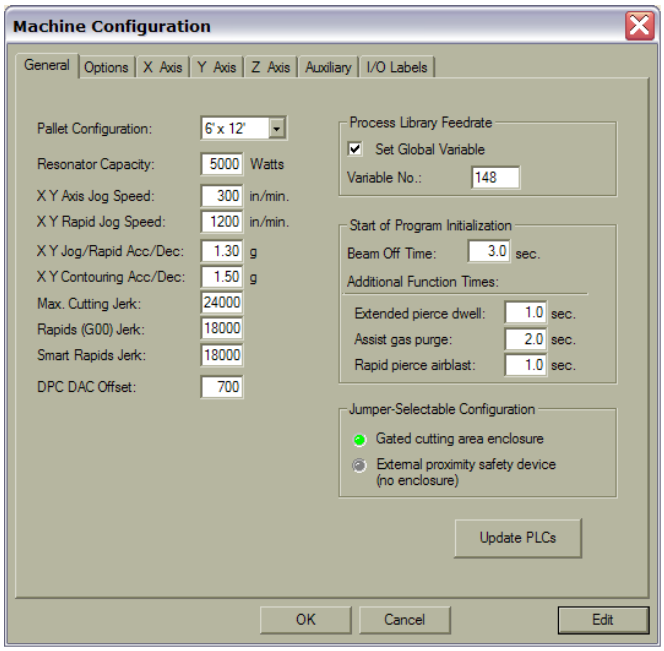
**Manager**: All values except those on the Options page can be edited.

**Operator**: Only a few parameters can be edited by operators. See the individual configuration page descriptions.

***Keyboard Shortcut:*** ***{ Alt, M, C } sequence***

## General Configuration

The General Configuration tab is used for viewing and modifying general machine configuration parameters. This page is always the first page displayed when the *Machine Configuration* window is opened.




*Pallet Configuration:* Specifies pallet size (and gantry travel). Choices are 4' x 8', 5' x 10', 6' x 12'2m x 4m or 8' x 20'.

*Resonator Capacity:* Specifies the nominal maximum power available from the laser resonator.

*X Y Axis Jog Speed:* The target speed when the Front Panel or Remote Station pushbuttons are used to jog the X-axis or Y-axis.

*X Y Rapid Jog Speed:* Specifies the jog speed for both the X- and Y-axes when the "RAPID TRAVERSE" button on the Front Panel is selected. This value must be greater than or equal to X/Y-axis Jog Speed.

*X Y Jog/Rapid Acc/Dec:* This parameter specifies the acceleration / deceleration rate for X-axis and Y-axis moves in Jog mode and during rapid (**G00**) moves.

*X Y Contouring Acc/Dec:* This parameter specifies the acceleration / deceleration rate for X-axis and Y-axis contouring moves (G1, G2, or G3).

*Max. Cutting Jerk:* Specifies maximum servo "Jerk" value used during contouring.

*Rapids (G00) Jerk:* Specifies the servo "Jerk" value used during rapid (**G00**) moves.

*Smart Rapids Jerk:* Specifies the servo "Jerk" value used during Smart Rapids moves.

*DPC DAC Offset:* To command pulse duty cycle, the Laser System control sends a signal to the laser resonator control. This factory-set parameter calibrates that signal to produce 100% duty cycle.

*Update PLCs:* The "Update PLCs" button starts the process of reloading PLC software in the PMAC motion controller. This process forces the **CNC/HMI** application to be shut down and restarted.

### Process Library Feedrate

*Set Global Variable:* Enables or disables the use of cutting speed in Process Parameter files. When checked, the contouring feedrate can be specified along with the other cutting parameters in Process Parameter windows. If not checked, the *Feedrate* parameter will not be visible in Process Parameter windows.

*Variable No.:* Specifies the number of the global variable used for the Process Parameter contouring feedrate. Valid variable numbers are from 100 to 999. If Set Global Variable is checked, the cutting speed specified in the Process Parameter file will be written to the global variable when the program commands a "**G89** P__" block. The "Variable No." field is visible only if "Set Global Variable" is checked.

*Note: This parameter can be edited by the Operator.*

### Start of Program Initialization

The *Start of Program Initialization* feature is intended to eliminate any inconsistencies that might occur on the first cut sequence following a long interruption in cutting. In the event that a cutting sequence has not occurred for a defined amount of time, the control will delay the next cut sequence briefly, allowing enough "initialization time" for the laser beam to stabilize, and to purge the assist gas and air blast lines. The cut sequence will then proceed as normal

*Beam Off Time:* The length of time for which the laser beam must be turned off between cuts to activate the **Start of Program Initialization** function. A value of zero disables Start of Program Initialization.

*Note: This parameter can be edited by the Operator.*

*Extended pierce dwell:* The amount of time by which the first pierce cycle time is automatically increased when the **Extended Pierce Cycle** function is activated. If no pierce cycle is programmed, a dwell of *Extra Cycle Time* seconds is executed after the beam turns on, before motion begins.

*Note: This parameter can be edited by the Operator.*

*Assist gas purge:* If the beam has been off for a period of time greater than *Beam Off Time*, the assist gas line will be purged for *Assist gas purge* time, before the next pierce is started.

*Note: This parameter can be edited by the Operator.*

*Rapid pierce airblast:* If the beam has been off for a period of time greater than *Beam Off Time*, an additional amount of time specified by *Rapid pierce airblast* will be added to the airblast duration of the next **Rapid Pierce** cycle.

*Note: This parameter can be edited by the Operator.*

### Jumper-Selectable Configuration

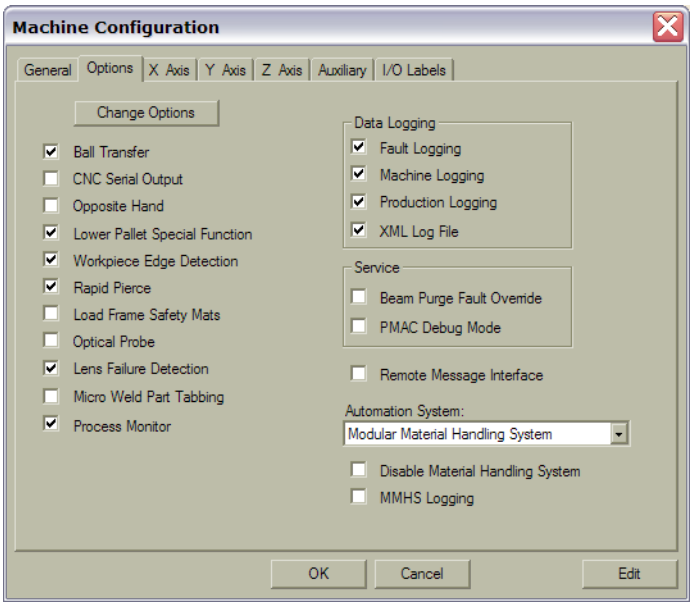
This area contains read-only indicators for machine configuration settings that are controlled by electrical input connections or jumpers. Enabled configuration items have green indicators.

*Gated cutting area enclosure:* Indicates this machine is equipped with a gated cutting area enclosure safety system.

*External proximity safety device (no enclosure):* Indicates this machine uses an external proximity safety system, such as an area scanner or safety mats, instead of a cutting area enclosure.

### Option Configuration

The Options tab contains check boxes to enable or disable various optional machine functions. An option is enabled if the corresponding box is checked.




*Change Options:* This button is only visible to the Administrator user, and prompts for the entry of an access code. The proper access code must be given before any machine options can be changed.

*Note: The "Edit" button will not be displayed until the "Change Options" button has been selected.*

*Note: The **Data Logging** and **Service** options do not require an access code, and be changed by the Manager user.*

*Ball Transfer:* This option indicates that the Laser System is equipped with optional ball transfer mechanism used to assist in moving material sheets on the pallets.

*CNC Serial Output:* This option enables the "DPRNT" option, configuring the control to output character strings to a remote computer via an additional serial port.

*Opposite Hand:* This option configures control software for opposite hand operation (the Operator Control Station is on the opposite side of the machine frame from the standard location).

*Lower Pallet Special Function:* This option allows laser cutting with a workpiece, such as rectangular tubing, on the lower pallet even though the top surface of the workpiece extends several inches above the pallet. Standard distance is 24 inches. Full travel for y may be implemented.

*Workpiece Edge Detection:* This option can be used to automatically shift and rotate the work coordinate system, effectively aligning the program with the workpiece.

*Rapid Pierce:* This option can be used to reduce piercing times for thick mild steel.

*Lens Failure Detection:* This optional feature incorporates an optical sensor to detect smoke and/or debris in the vicinity of the cutting lens. If possible damage is detected, the beam will be turned off and a system alarm will be reported.

*Micro Weld Part Tabbing:* This option can be used in conjunction with the Modular Material Handling System automation to prevent parts cut from thick material from dropping out of the sheet during transport.

*Process Monitor:* This option enables the optional Process Monitor System, which uses measurements of light intensity and plasma conductance to improve cutting performance.

### Data Logging Options

*Fault Logging:* Enables automatic logging of system alarms and laser alarms to the text file "FAULT.LOG" in folder "\\Cnclsr32\".

*Machine Logging:* Enables automatic logging of various statistical and configuration data to the text file "MACHINE.LOG" in folder "\\Cnclsr32\".

*Production Logging:* Enables automatic logging of program execution history data to the text file "PRODUCT.LOG" in folder "\\Cnclsr32\".

*XML Log File:* Enables automatic daily recording of all Fault, Machine, and Product log data to one file in ".xml" format. The file name indicates the date: "yyyymmdd.XML"; where "yyyy" is the year, "mm" is the month (01 to 12) and "dd" is the date (01 to 31). The daily ".xml" files are in folder "\\Cnclsr32\Logs\". This must be enabled if the remote web dashboard is used on computers on the same network. See remote message interface.

### Service Options

*Beam Purge Fault Override:* Prevents the "Beam purge pressure fault" alarm from being reported, so the machine can be cycled without purge gas. The FYI message "Beam purge loss-of-pressure alarm disabled" will be displayed when this option is enabled.

*PMAC Debug Mode:* Enables the display of diagnostic information output by the PMAC servo controller to the *PMAC Interface Console* window. This is used only by CINCINNATI INCORPORATED personnel for diagnostic purposes.

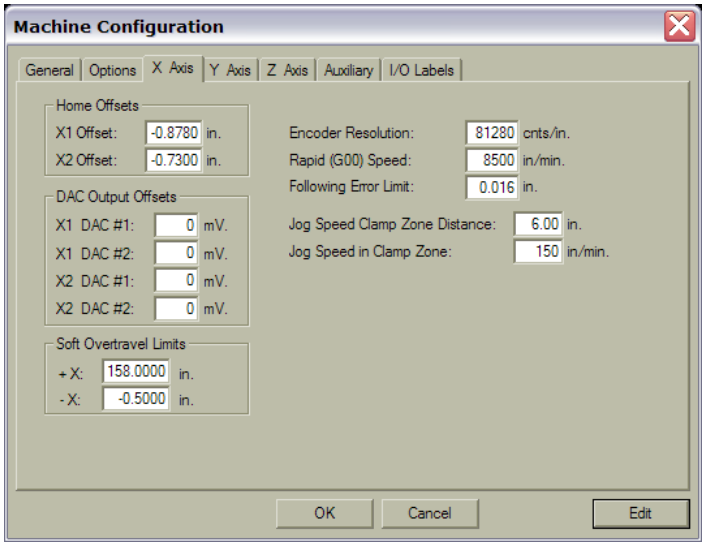
### Automation System Options

*Automation System:* This setting configures the Laser System software to interface with optional pallet loading equipment. The Options page (in Edit mode) displays the Automation System options in a pull-down menu. For a detailed description of the Automation system options, contact CINCINNATI Laser Service.

*Remote Message Interface:* This setting configures the Laser System software to communicate with other software provided by the user. The software applications can exchange messages regarding which program to run, machine status, etc. For a detailed description, contact CINCINNATI Laser Service.

## X and Y Axis Configuration

The X Axis Configuration and Y Axis Configuration tabs are used to view and modify parameters pertaining to the X-axis and Y-axis servo systems. Two nearly identical pages are provided, one for X-axis parameters and one for Y-axis parameters.




*Home Offset:* Specifies the distance from Home reference mark where the axis will come to rest after completing the Axes Home operation.

*DAC Output Offsets:* The DC offsets of the motion controller command signals. These values are set during the initial setup and calibration of the servo system.

*Soft Overtravel Limits:* Specifies the positions, in machine coordinates, beyond which controlled axis motion is not allowed. If an axis reaches a position less than the negative overtravel limit or greater than the positive overtravel limit, axis motion in the same direction is disabled and an alarm is reported.

*Rapid (G00) Speed:* The desired (maximum) speed to be achieved during rapid (**G00**) moves.

*Following Error Limit:* The maximum magnitude the servo following error can attain before the "Fatal following error" alarm is reported.

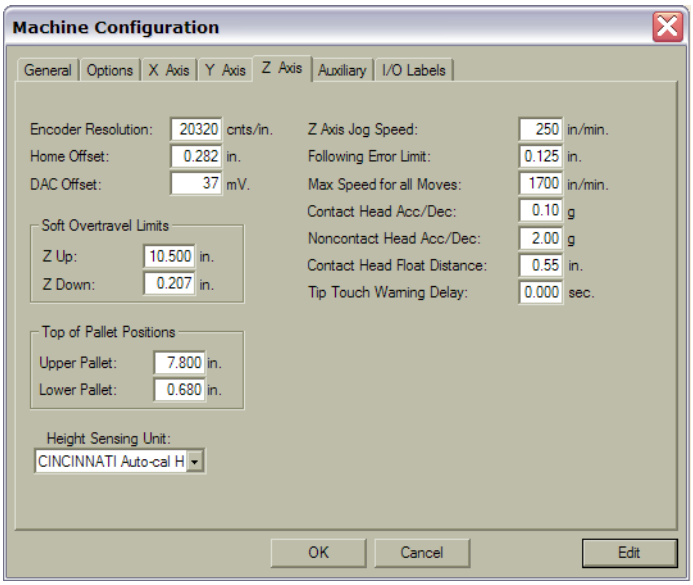
*Jog Speed Clamp Zone Dist.:* The jog speed clamp zone is an area inside the perimeter of the X-axis and Y-axis travel limits. This parameter specifies the distance from the start of the clamp zone to the nominal travel limit for each axis. When jogging toward the travel limit in this zone, the jog feedrate is automatically limited to the Jog Speed in Clamp Zone parameter. Outside this zone, the jog speed shown in the General Machine Configuration tab is used.

*Jog Speed in Clamp Zone:* The speed to which the axis is automatically limited when jogging toward the travel limit while inside the rapid clamp zone. This value must be less than or equal to X/Y-axis Jog Speed on the General Machine Configuration tab.

*Y-Travel Limit for Lower Pallet Special Function Mode (M43):* (Y axis page only) When using the Lower Pallet Special Function, the Y-axis travel range is limited for safety purposes. Whenever **M43** is active, this value replaces the +Y Soft Overtravel Limit as the positive Y-axis travel limit. Standard distance is 24 inches. Full travel for y may be implemented.

## Z Axis Configuration

The Z Axis Configuration page is used for viewing and modifying parameters pertaining to the Z-axis servo system.




*Soft Overtravel Limits:* Specifies the positions, in machine coordinates, beyond which controlled axis motion is not allowed. If the Z-axis reaches a position less than the negative overtravel limit or greater than the positive overtravel limit, axis motion in the same direction is disabled and an alarm is reported.

*Top of Pallet Positions:* Specifies the positions, in absolute Z-axis coordinates, corresponding to the top of the upper and lower pallets when material supports are new.

*Home Offset:* Specifies the Distance from Home reference mark where the axis will come to rest after completing the Axes Home operation.

*DAC Offset:* The DC offset of the Z-axis D/A converter / amplifier. This value is set during the initial setup and calibration of the Z-axis servo system.

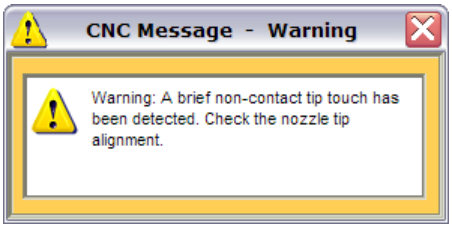
*Z-axis Jog Speed:* The target speed when the "Front Panel" or "Remote Station" pushbuttons are used to jog the Z-axis.

*Following Error Limit:* The maximum magnitude the Z-axis servo following error can attain before the "Fatal following error" alarm is reported.

*Max Speed for all Moves:* The target speed for all programmed Z-axis moves.

*Noncontact Head Acc/Dec:* The rate of acceleration and deceleration for all Z-axis moves when the noncontact cutting head is used.

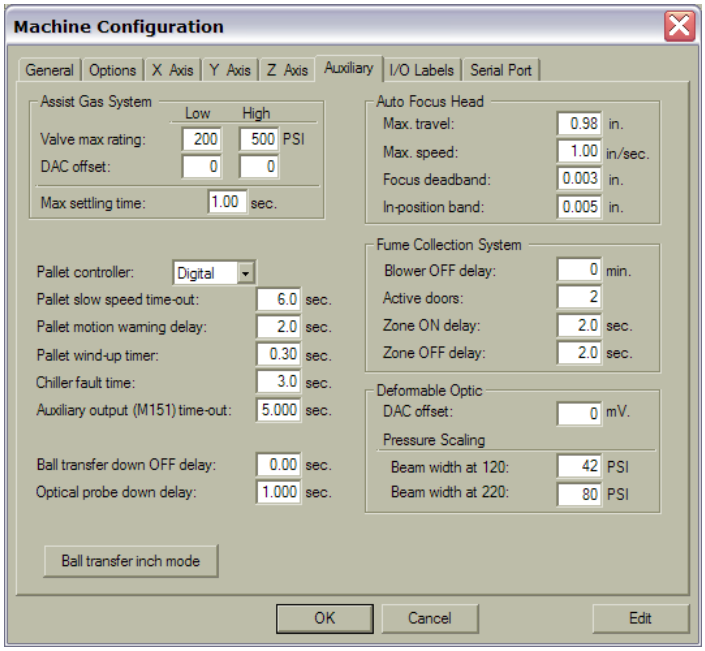
*Tip Touch Warning Delay:* When a brief noncontact tip touch condition occurs, the software will display this warning message:




If the tip touch condition clears before the **Tip Touch Warning Delay** expires, the software will not stop the program. If the tip touch condition lasts longer than the Warning Delay, then a "Noncontact Z-tip touch" system alarm will be reported and program execution will be terminated.

## Auxiliary Configuration

The Auxiliary Configuration tab is used for viewing and modifying configuration parameters for auxiliary functions and subsystems.




*Pallet Controller:* This parameter configures the Laser System control to interface with the specified type of pallet drive.

*Pallet Slow Speed Time-out:* When the pallet enters the slow speed phase, the control allows the pallet this much time to reach position, before it reports the "Pallet motion timed out" alarm.

*Pallet Motion Warning Delay:* Some Laser Systems use a signal (a horn or light) to warn operators before moving the pallets. This parameter determines how long the software activates the warning signal before it moves the pallets. The control software starts the warning signal when the program or the operator requests pallet motion. The warning signal continues until pallet motion stops.

*Pallet Wind-Up Timer:* After receiving the signal that a pallet reached the commanded position, the pallet drive continues to pull on the pallet for a short time, applying tension to hold the pallet against the stop. The configuration parameter specifies this "wind-up" time.

*Chiller Fault Time:* A Chiller Fault condition occurs when the laser control software activates the output to start the chiller but does not receive a return signal confirming that the chiller started. This parameter determines how long the software waits between recognizing the Chiller Fault condition and activating the Chiller Fault system alarm. If the chiller starts during that time, the software does not activate the alarm. The time delay is a configuration parameter because different chiller models may require different starting times.

*Auxiliary Output (M151) Time-out:* When an **M151** code is commanded in an NC Program, program execution pauses and waits for the corresponding **M151** customer input to turn on. This parameter specifies the maximum time that the control will wait for the **M151** customer input, at which point an **M151** alarm will be triggered.

*Ball Transfer Down OFF Delay:* Specifies additional motor rotation time after the ball transfer down switch has been tripped. This value is used to fine-tune the final stopping position of the ball transfer, and ensure that the lower pallet can pass over the mechanism.

*Ball Transfer Inch Mode:* Specifies a special "manual adjustment" mode for maintenance purposes. When this button is depressed, the ball transfer will decelerate to a stop whenever the remote pendant buttons are released. Normal operation dictates that the ball transfer move all the way up / down with any momentary button press. This mode allows for manual adjustment of the proximity switches, without risk of overtravel.

### Assist Gas System

*Valve Max Rating:* The range of the assist gas pressure valve, for example, the pressure output expected when the maximum command signal is input to the valve.

*DAC Offset:* The assist gas pressure controller receives a voltage signal from the Laser System control. This parameter adjusts the signal for a commanded pressure. The gas pressure calibration procedure sets this parameter.

*Note: The two parameters above each have a **Low** and **High** setting. The **High** settings apply to the higher pressure rated valve, which is used for Nitrogen assist gas. The **Low** settings apply to the lower pressure rated, higher accuracy valve used for Oxygen and Air assist gas.*

*Max Settling Time:* When the program changes the commanded assist gas pressure, the control allows the assist gas system this much time to respond before it reports an "Assist gas valve failed to maintain target pressure" alarm.

### Auto Focus Head

See the **AUTO FOCUS CUTTING HEAD** section for Auto Focus Head configuration parameter descriptions.

### Fume Collection System

*Blower OFF delay:* When a program ends, the fume blower remains on for the time specified by this parameter.

The Auxiliary page displays the next three parameters only when Pallet Configuration on the General Configuration page is *8'x 20'*. The 8' x 20' cutting area uses a fume collection system with multiple exhaust sections controlled with separate damper gates. The software only opens the gates for exhaust sections near the cutting head position.

*Active Doors:* This parameter specifies the number of fume exhaust section gates that the software will keep open at the same time.

*Zone On Delay:* If the cutting head moves into the area of a closed exhaust section for this length of time, then the software opens the gate for that section.

*Zone Off Delay:* If the cutting head moves outside the area of an open exhaust section for this length of time, then the software closes the gate for that section.

### Deformable Optic

The Deformable Optic system is used to modify beam size in order to optimize cutting speeds and quality, and minimize pierce times for materials of varying thickness and type. A cut and pierce *Beam Size* setting is preset in each Process Parameter file.

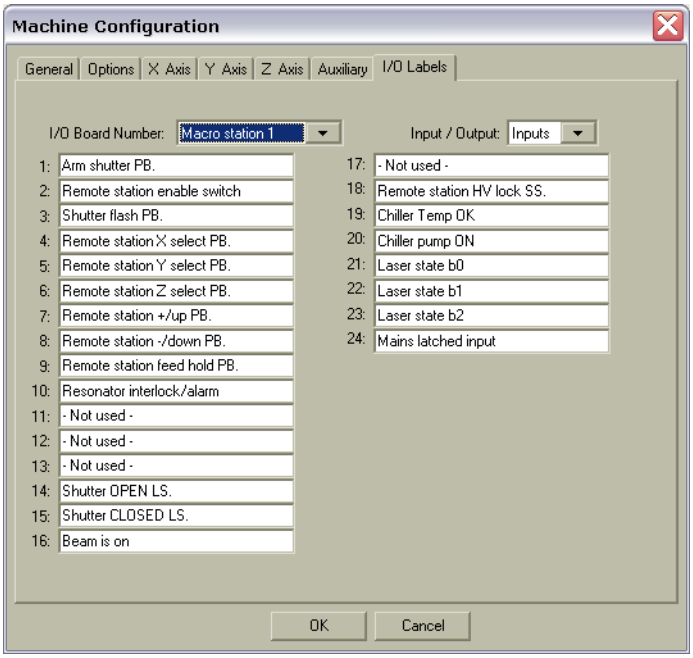
*DAC Offset:* The DC offset of the deformable optics pump D/A converter / amplifier. This value is set during the initial setup and calibration of the deformable optic system.

*Pressure Scaling, Beam Width at 120:* The first calibration point for the Deformable Optic pump circuit. This value is set by factory or service personnel, after collecting the appropriate beam diameter information using a beam monitoring device.

*Pressure Scaling, Beam Width at 220:* The second calibration point for the Deformable Optic pump circuit. This value is set by factory or service personnel, after collecting the appropriate beam diameter information using a beam monitoring device.

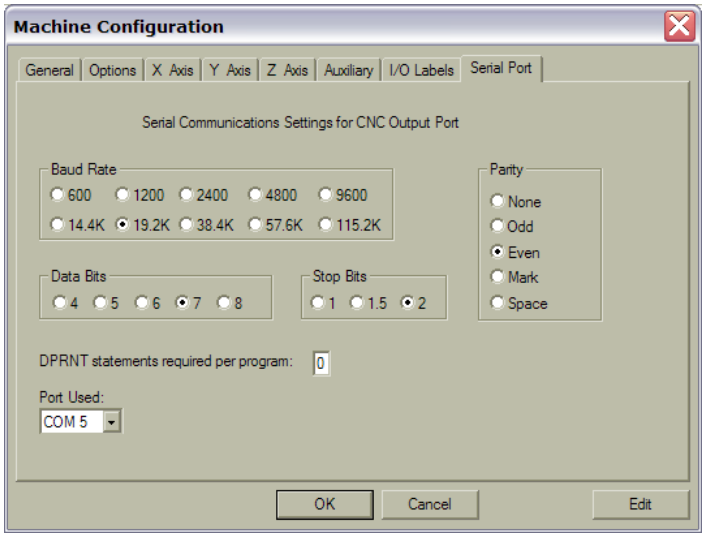
## I/O Labels Configuration

The I/O Labels Configuration tab is used for viewing and modifying the machine I/O description labels. Both input and output labels are displayed for all machine I/O boards. These I/O labels are displayed in the I/O Monitor windows when a specific I/O state edit control is selected, and in the I/O Watch Window when the selection dialog is used to configure a specific watch point. The I/O Labels Configuration tab can be used to add or change description labels for custom I/O connections.




## Serial Port Configuration

The Serial Port Configuration tab is used for viewing and modifying the serial communications settings for the Serial Output Port. The Serial Port Configuration tab is visible only if the 'CNC Serial Output' box is checked on the Options Machine Configuration tab. If that option is not enabled, the serial output port cannot be configured. Baud rate, data bits used, stop bits used and parity type can be configured. '**DPRNT** statements required per program' specifies the minimum number of **DPRNT** statements a program must contain in order to be executed without errors. 'Port Used' specifies which COM port is used for the **CNC Serial Output** function. For most Laser Systems, this should be set to "COM 5".



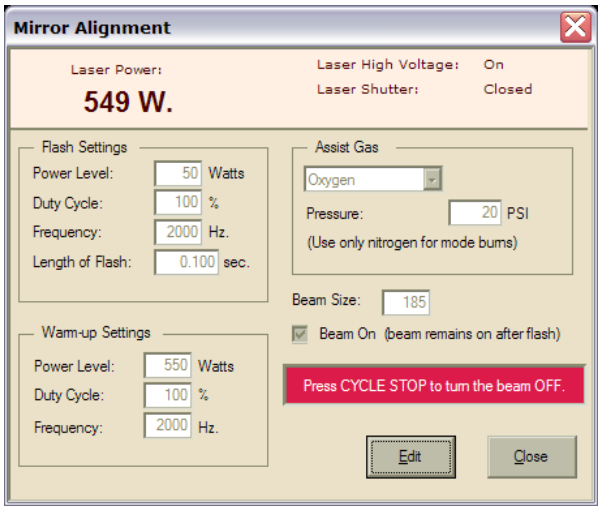
## MIRROR ALIGNMENT

**Maint | Mirror Alignment** opens the *Mirror Alignment* window. As long as this window is open, the Laser System will be in Mirror Alignment mode. This mode is a "special" operating mode that is used to perform mirror alignment and other resonator maintenance functions. In the Mirror Alignment mode, the laser beam is activated directly by the "CYCLE START" button on the Operator Control Station front panel, or the "SHUTTER FLASH" button on the Remote Station. See **BEAM DELIVERY ALIGNMENT** in **SECTION 9 – MAINTENANCE AND ADJUSTMENTS** of the **Laser System Operation, Safety, and Maintenance Manual** for mirror alignment procedures.

**! WARNING !**

**THE ALIGNMENT PROCEDURE IS TO BE PERFORMED ONLY BY CINCINNATI TRAINED SERVICE PERSONNEL. DO NOT ATTEMPT TO ALIGN THE BEAM DELIVERY SYSTEM WITHOUT PRIOR TRAINING, PROPER TOOLS, AND HAVING READ AND UNDERSTOOD THIS PROCEDURE. FAILURE TO COMPLY WITH THIS WARNING COULD RESULT IN SERIOUS INJURY OR DEATH TO YOU OR OTHERS.**

*Note: When the **Mirror Alignment** window is open, the Laser System is in a diagnostic mode used only for machine maintenance. Normal machine operation such as program execution is disabled while the Mirror Alignment mode is active.*



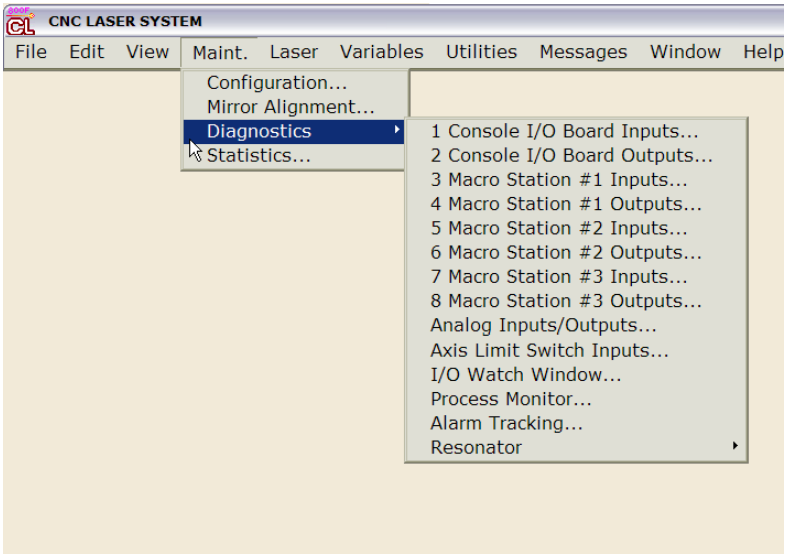

From the *Mirror Alignment* window, the laser can be configured for manual operation by setting the *Power Level*, *Frequency*, *Duty Cycle*, *Assist Gas Pressure*, and *Length of Flash* time. The 'Beam On' check box configures the control to keep the laser beam on before and after the shutter flash. This operating mode simulates normal operation by testing beam alignment with the laser resonator in a warmed-up condition. When 'Beam On' is checked, the *Mirror Alignment* window has separate Warm-up Settings for *Power*, *Duty Cycle*, and *Frequency*.

To change any of the *Mirror Alignment* parameters, select the "Edit" button, make the desired changes, and then select "End Edit".

***Keyboard Shortcut:*** ***{ Alt, M, M } sequence***

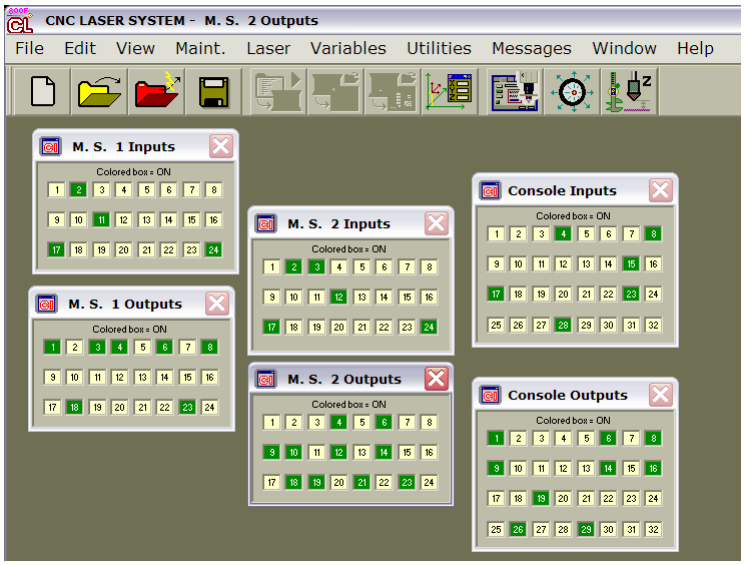
## DIAGNOSTICS

Several diagnostic windows are accessible via the *Diagnostics* fly-out submenu, which appears when the **Maint | Diagnostics** command is selected.




### Input / Output Monitor Windows

The *I/O Monitor* windows are used to view the states of any discrete inputs or outputs used by the Laser System. The monitor windows are activated from the **Maint | Diagnostics** submenu by selecting the desired menu item or by using the indicated shortcut keys (underlined characters in the submenu). More than one *I/O Monitor* window can be open at the same time. *I/O Monitor* windows are read-only windows; none of the displayed information can be modified by the user with the trackball or keyboard.

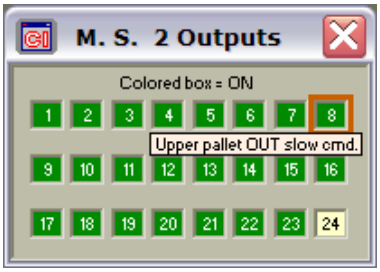



Each *I/O Monitor* window contains color-coded state indicators, one for each input or output of the corresponding Macro Station or Console I/O board. The state indicators are arranged in rows of 8; the terminal number is displayed inside each 'State Indicator' text box. If the corresponding input or output is on, the state indicator is colored green with a white terminal number. Otherwise, the state indicator is colored yellow with a black terminal number. Once open, *Macro Station* or *I/O Monitor* windows are automatically updated to reflect any changes in the displayed input or output states.

***Keyboard Shortcut:*** ***{ Alt, M, D, 1 } , { Alt, M, D, 2 } … { Alt, M, D, 8 } sequence***

### Input / Output Labels

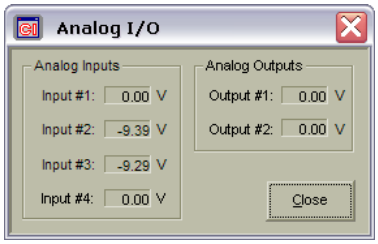
A brief description for each input or output point can be viewed from the *I/O Monitor* windows by selecting an "I/O state" indicator, which will cause a small window to open just below the cursor.




The window text is the label of the corresponding input or output. The selected state indicator will be surrounded by an orange highlight, indicating that the label corresponds to that terminal. Selecting outside the state indicator will close the label window.

### Analog Inputs / Outputs

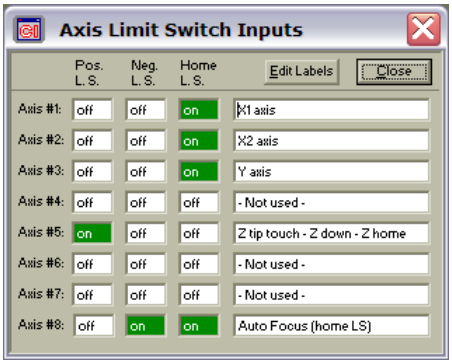
The *Analog I/O* window is a diagnostic tool for monitoring the values of all control system analog inputs and outputs. Once this window is opened, the displayed values are continuously updated. This window is for monitoring purposes only. None of the displayed values can be changed from the window.




***Keyboard Shortcut:*** ***{ Alt, M, D, A } sequence***

### Axis Limit Switch Inputs

The *Axis Limit Switch Inputs* window uses 24 color-coded state indicators to display the states of 3 limit switches for each of the 8 motion controller servo axes. There is an indicator for the positive limit switch, negative limit switch and home limit switch for all 8 axes. If a limit switch is tripped, its indicator will display "on", using white text on a green background. Otherwise, the indicator will display "off" in black text on a white background. Once open, this window is automatically updated to reflect any changes in the axis limit switch states.

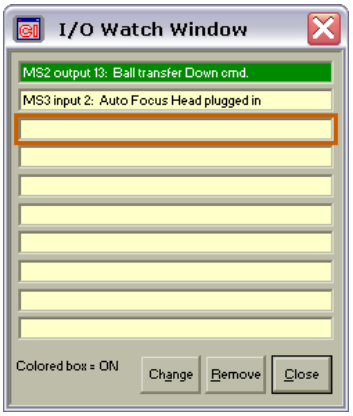

Axis Limit Switch Inputs
Pos. L.S.
Neg. L.S.
Home L.S.
Edit Labels
Close
Axis #1: off off on X1 axis
Axis #2: off off on X2 axis
Axis #3: off off on Y axis
Axis #4: off off off - Not used -
Axis #5: on off off Z tip touch - Z down - Z home
Axis #6: off off off - Not used -
Axis #7: off off off - Not used -
Axis #8: off on on Auto Focus (home LS)

A label field is provided to give a logical description of how each axis is currently used. Each label field is a text box, which users logged-in as "Administrator" can edit by selecting the "Edit Labels" button.

***Keyboard Shortcut:{ Alt, M, D, L } sequence***

## I / O Watch Window

The *I/O Watch Window* allows any 10 user-selectable I/O points to be monitored together in the same window. The *I/O Watch Window* contains 10 read-only text boxes, which, similar to the *I/O Monitor Windows*, are used as color-coded state indicators. The text boxes display the board number, terminal number and a brief description of the corresponding I/O point. If the indicated input or output is on, the state indicator is colored green with white text. Otherwise, the state indicator is colored yellow with black text. Once open, the *I/O Watch Window* is automatically updated to reflect any changes in the displayed input or output states.



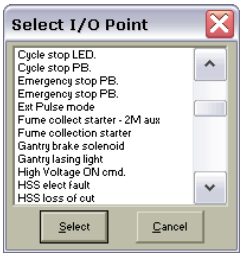

Initially, all text boxes are blank. Use the "Change" and "Remove" buttons to configure the *Watch* window. Select an I/O point for monitoring as follows:

1. Using the touchscreen or trackball button, select the text box to be used as the state indicator. The selected box will be highlighted with an orange outline to indicate the selection.

2. Select the "Change" button. This will open a small window containing a scrolling list of description labels for all discrete inputs and outputs. This dialog box will contain two buttons: "Select" and "Cancel".

3. Scroll the I/O label list to find the label of the input or output point to be monitored. Select the I/O label (the selection is highlighted with blue background).

4. Click the "Select" button to add the I/O point to the *I/O Watch Window*. Click "Cancel" to abort the selection. Either "Select" or "Cancel" will close the dialog box.

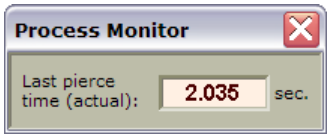



To stop monitoring an I/O point in the *I/O Watch Window*, first select the text box of the I/O point, and then select the "Remove" button. The selected state indicator text box will be cleared.

***Keyboard Shortcut: { Alt, M, D, W } sequence***

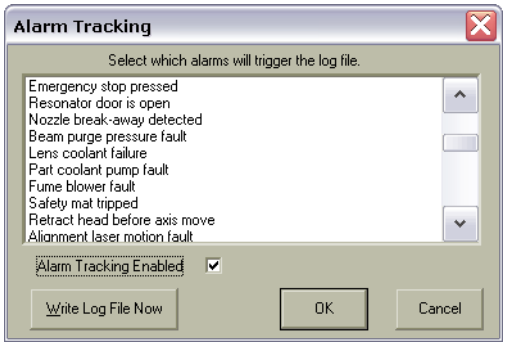
## Process Monitor Window

For lasers equipped with the optional **Process Monitor** system, this window displays actual pierce times.



### Alarm Tracking

Use the *Alarm Tracking* window to configure the **Alarm Tracking** diagnostic function. Alarm Tracking allows diagnostic information to be automatically captured and saved to a disk file after a selected system alarm occurs.



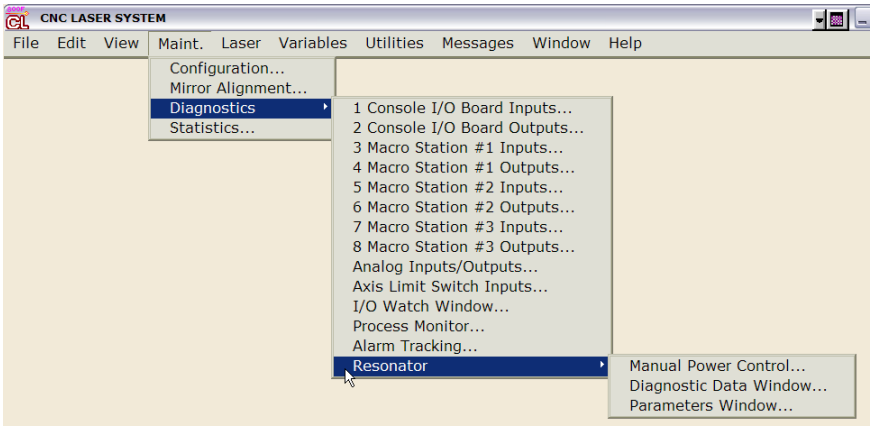

To enable alarm tracking, select any number of system alarms to trigger diagnostic data recording, make sure the Alarm 'Tracking Enabled' box is checked, and then select "OK". If one of the selected system alarms occurs, a short diagnostic file will be written to the hard disk. The name of this file will be unique so that old data is not overwritten. The "Write Log File Now" button causes the diagnostic file to be written immediately without waiting for a system alarm. This button is disabled if Alarm Tracking is not enabled.

*Note: Alarm Tracking is a troubleshooting aid. While it is enabled, large numbers of files will be written to the hard drive. Do not leave Alarm Tracking enabled on a permanent basis during normal operation.*

***Keyboard Shortcut: { Alt, M, D, T } sequence***

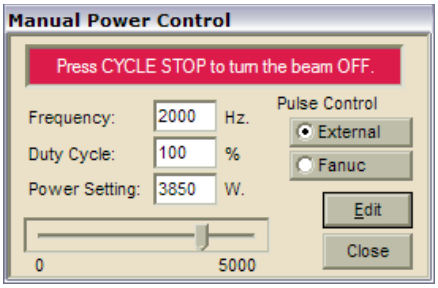
## RESONATOR

Use the **Maint. | Diagnostics | Resonator** command to display a submenu of diagnostic functions for the laser resonator.




### Manual Power Control

Use the **Manual Power Control** command to open the *Manual Power Control* window. As long as this window is open, the Laser System will be in Manual Power Control mode. This is a "Special Operating Mode" used for setup and maintenance purposes. In Manual Power Control mode, the laser beam can be turned on manually with the shutter closed by pressing the "CYCLE START" button on the Operator Control Station front panel.




To change the settings, first select the "Edit" button. The "Edit" button will change to "Update". Set the desired power level using the slider control or by entering a new value into the "Power Setting" field. After the desired power level and pulse parameters are set, use the "Update" button to transfer the power settings to the resonator control. Press the "CYCLE START" button to turn on the laser beam.

*Note: The shutter remains closed while the Laser System is in Manual Power Control mode.*

The Pulse Control buttons are used to select the source of the pulse control signal.

*External:* The CINCINNATI pulse control system controls the resonator power. This is the default setting.

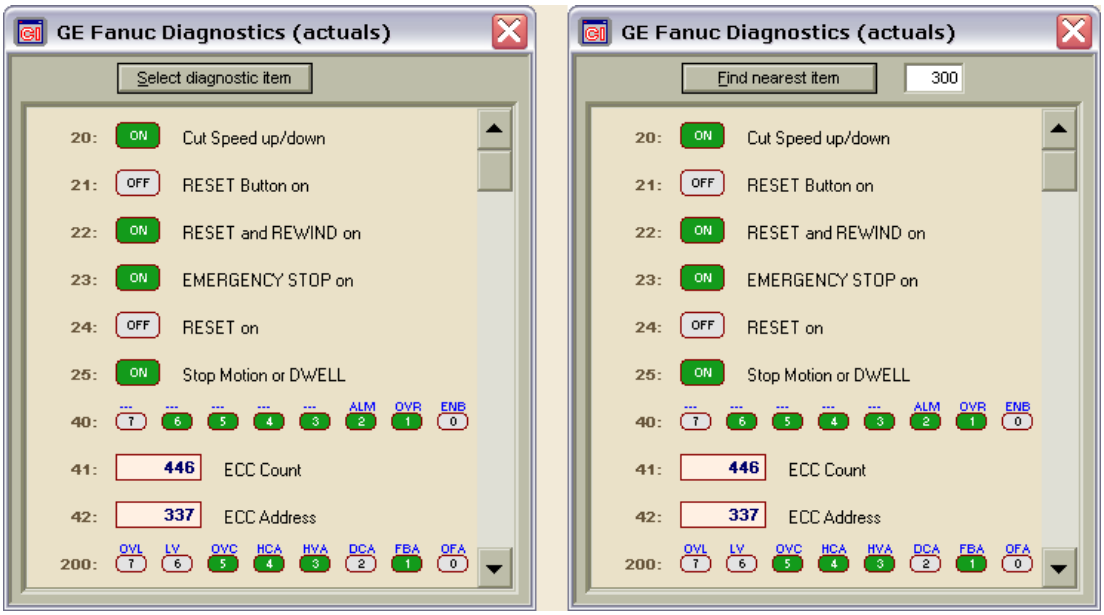
*Fanuc:* The internal pulse generator (with power regulation) controls the resonator power.

***Keyboard Shortcut:*** ***Ctrl + M or { Alt, M, D, R, M } sequence***

## Diagnostic Data Window

The *Diagnostics Data* window allows displays the read-only "Diagnostics" data within the resonator control. See the CNC manuals for Diagnostics data descriptions.

*Select diagnostic item:* Use this button to quickly display a particular diagnostic item number. Selecting the button opens an edit field and changes the button text to "Find nearest item". Type the desired item number in the edit field and then select "Find nearest item". The software will scroll the window to display the specified item. If the specified item does not exist, the window will display the nearest matching diagnostic items.


GE Fanuc Diagnostics (actuals)
Select diagnostic item
20: ON Cut Speed up/down
21: OFF RESET Button on
22: ON RESET and REWIND on
23: ON EMERGENCY STOP on
24: OFF RESET on
25: ON Stop Motion or DWELL
40: ... ... ... ... ... ALM OVR ENB
7 6 5 4 3 2 1 0
41: 446 ECC Count
42: 337 ECC Address
200: OVL LV OVC HCA HVA DCA FBA OFA
7 6 5 4 3 2 1 0
GE Fanuc Diagnostics (actuals)
Find nearest item
300
20: ON Cut Speed up/down
21: OFF RESET Button on
22: ON RESET and REWIND on
23: ON EMERGENCY STOP on
24: OFF RESET on
25: ON Stop Motion or DWELL
40: ... ... ... ... ... ALM OVR ENB
7 6 5 4 3 2 1 0
41: 446 ECC Count
42: 337 ECC Address
200: OVL LV OVC HCA HVA DCA FBA OFA
7 6 5 4 3 2 1 0

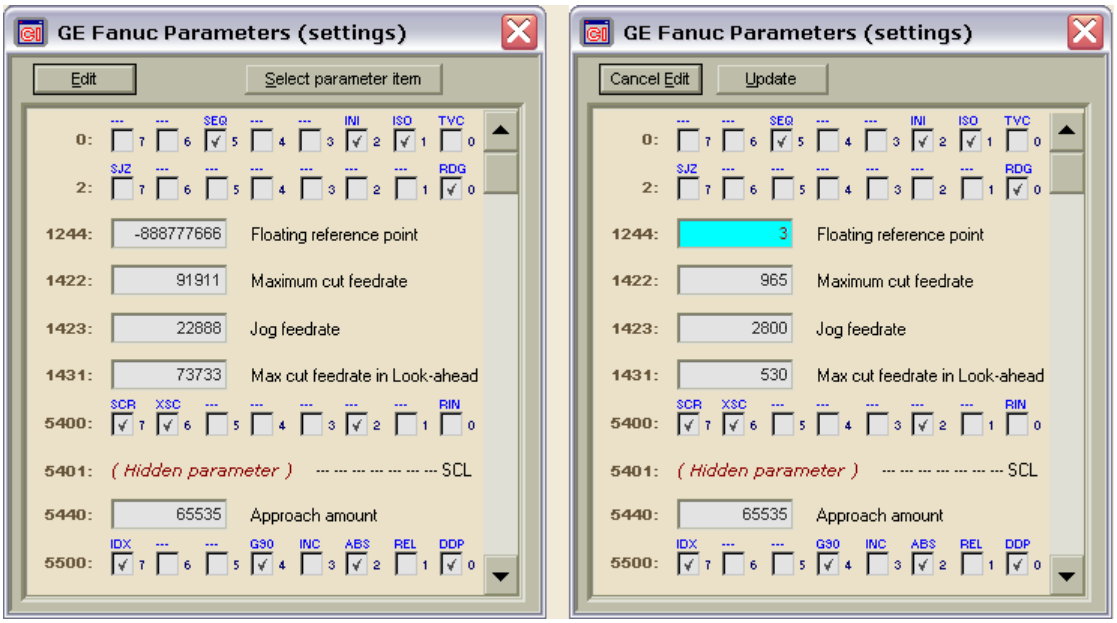
***Keyboard Shortcut:*** ***{ Alt, M, D, R, D } sequence***

## Parameters Window

The *Parameters Window* provides access to the "Parameter" settings used by the resonator control. See the CNC manuals for descriptions of the Parameters.

*Select parameter item:* Use this button to quickly display a particular parameter number. Selecting the button opens an edit field and changes the button text to "Find nearest item". Type the desired parameter number in the edit field and then select "Find nearest item". The software will scroll the window to display the specified parameter. If the specified parameter is not used by the control, the window will display the nearest parameter.

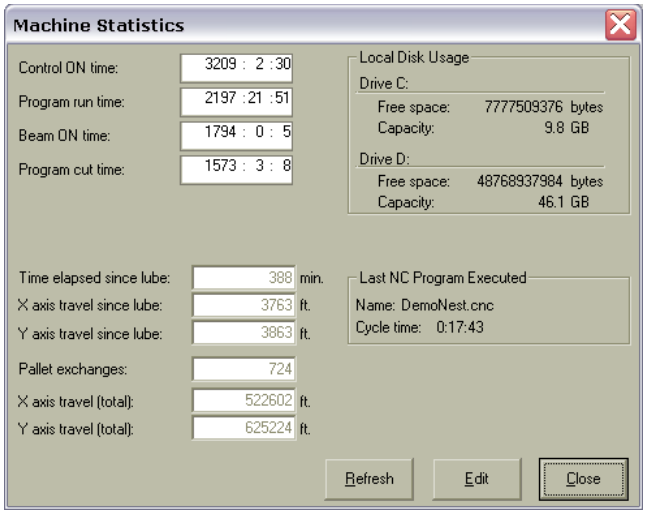
*Edit:* Selecting this button enables Parameter Editing mode, changes the button text to "Cancel Edit" and adds a button labeled "Update". When a parameter field is selected for editing, its background color changes to highlight the action. After editing, select the "Update" button to apply the new settings, or the "Cancel Edit" button to ignore the changes.


GE Fanuc Parameters (settings)
Edit
Select parameter item
0: --- --- SEQ --- --- INI ISO TVC
2: SJZ --- --- --- --- --- --- RDG
1244: -888777666 Floating reference point
1422: 91911 Maximum cut feedrate
1423: 22888 Jog feedrate
1431: 73733 Max cut feedrate in Look-ahead
5400: SCR XSC --- --- --- --- --- RIN
5401: ( Hidden parameter ) --- --- --- --- --- --- --- SCL
5440: 65535 Approach amount
5500: IDX --- --- G30 INC ABS REL DDP
GE Fanuc Parameters (settings)
Cancel Edit
Update
0: --- --- SEQ --- --- INI ISO TVC
2: SJZ --- --- --- --- --- --- RDG
1244: 3 Floating reference point
1422: 965 Maximum cut feedrate
1423: 2800 Jog feedrate
1431: 530 Max cut feedrate in Look-ahead
5400: SCR XSC --- --- --- --- --- RIN
5401: ( Hidden parameter ) --- --- --- --- --- --- --- SCL
5440: 65535 Approach amount
5500: IDX --- --- G30 INC ABS REL DDP

***Keyboard Shortcut:*** ***{ Alt, M, D, R, P } sequence***

## STATISTICS

The **Maint. | Statistics** command opens the *Machine Statistics* window. Use this window to view statistical information pertaining to the Laser System. All data values are initially read-only. However, most values can be changed (for resetting purposes) by Administrator and Manager users.




*Control ON time:* Total accumulated run time for the Laser System **CNC/HMI Software** program (hours : minutes : seconds).

*Program Run time:* Program execution time. This counter begins when "CYCLE START" is pressed, and stops when program execution is terminated by "CYCLE STOP", **M30**, system alarm, etc.

*Beam ON time:* The cumulative time the laser beam has been energized (hours : minutes : seconds). This value includes the time when the beam was on during warm-up/calibration, mirror alignment, lens centering and manual laser power control operations.

*Program Cut time:* The cumulative time the laser beam has been on while executing a program (hours : minutes : seconds). Warm-up/calibration, mirror alignment, lens centering, etc. times are not included.

*Time Elapsed Since Lube:* Elapsed time since the X-axis and Y-axis linear bearings were lubricated. Change this value to zero after manually lubricating the bearings. This time value only increases while a program is executing.

*X / Y-axis Travel Since Lube:* The linear distance each axis has traveled since the bearings were lubricated. Change these values to zero after manually lubricating the bearings.

*Number of Pallet Exchanges:* The number of times either of the following has occurred (in Auto or Jog mode):

- The upper pallet moved to the cut position AND the lower pallet moved to the load position.
- The upper pallet moved to the load position AND the lower pallet moved to the cut position.

*X / Y-axis Travel:* The cumulative linear distance each axis has traveled.

*Local Disk Capacity:* The total storage capacity of the Laser host PC hard disk drive.

*Local Disk Free Space:* The number of free bytes on the Laser host PC hard disk drive.

*Last NC Program Name:* The name of the last cutting program loaded for execution.

*Last NC Program Cycle Time:* Time required to execute the last program. This time measurement starts when "CYCLE START" is pressed and ends when **M30** or **M02** is executed in the program.

### Button Operation

*Edit:* This button allows specific users to reset counter values. Administrator and Manager users can edit all statistics settings except *Last NC Program,* and *Local Disk Usage*.

*Close:* This button closes the Machine Statistics window. If any values have changed, a message box will open, asking if the changes should be saved (user response: "Yes", "No" or "Cancel"). Select "Yes" to save the changes or "No" to discard the changes. Selecting "Cancel" aborts the Close operation.

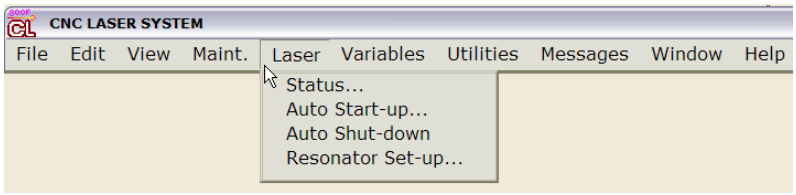
*Refresh:* Selecting this button will cause the most recent statistics to be collected and displayed.

*Note: The control does not automatically update any of the displayed statistics after the window is opened. The user must select the "Refresh" button or close/re-open the dialog box to see the current statistics.*

***Keyboard Shortcut:*** ***{ Ctrl + A} or { Alt, M, S} sequence***

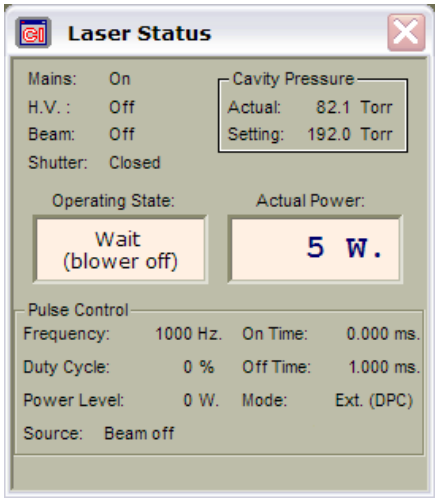
## LASER MENU

Use the *Laser* menu for access to resonator-related status information and functions.




### LASER STATUS MONITOR

**Laser | Status** opens the *Laser Status Monitor* window, which displays real-time resonator status information. The information in this window is for viewing only. Laser parameters cannot be edited from the *Laser Status Monitor* window.



*Operating State:* This field indicates one of the following operating states for the laser resonator:

- *Power Off* The resonator power is turned off.
- *Purged* The laser gas in the resonator has been replaced with purge gas; the resonator is shut down.
- *Purging* Gas purge is in progress (filling discharge tubes to atmospheric pressure with laser purge gas).
- *Wait (blower off)* Waiting (75 sec) for the turbo blower to stop, following a **Mains Off** command.
- *Evacuating* Start-up evacuation is in progress, after a **Mains On** command.
- *Wait (blower on)* Waiting for turbo blower target RPM and "discharge ready" pressure; Starting gas pressure control.
- *H.V. Ready* Discharge preparation is complete; Press the "High Voltage" button to start the laser.
- *Wait (H.V. off)* Waiting for return to "discharge ready" pressure in response to **HV OFF** command.
- *Start Discharge* Starting the resonator base discharge in response to **HV ON** command.
- *Wait (discharge)* Waiting for the base discharge state.
- *Calibrating* Laser power calibration is in progress. After the base discharge state is achieved in response to an **HV ON** command, the resonator control starts an automatic power calibration process in which a series of varying power levels is executed, and the resultant actual power is monitored.
- *Run* Oscillation preparation is complete; The laser is ready for **Beam On** commands.
- *Leak Check* Automatic leak check is in progress.
- *Laser Alarm* A resonator control system alarm is active

*Pulse Control Mode:* The resonator can operate in one of these pulse modes:

- *Ext. (fixed)* The pulse control signal is a constant setting from a source external to the resonator (i.e., from the Laser System control).
- *Ext. (DPC)* The external pulse control signal is from the **Dynamic Power Control** function in the Laser System control.
- *Internal* The pulse control signal originates in the resonator control system (for example, during Calibration).

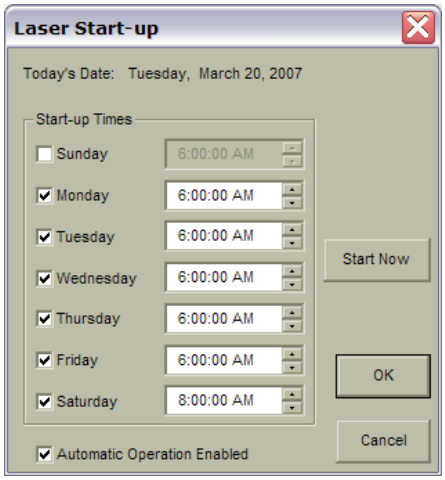
*Pulse Control Source:* Indicates the source of the pulse settings. The "Source" field indicates if the pulse control settings are from a **Maint. | Utility** function or from an NC cutting program. Four sets of pulse parameters are available for use by cutting programs. Each set contains separate *Cut*, *Pierce*, *Rapid Pierce*, and *Aux* parameters. During program execution, the "Source" field indicates which parameter set is currently being used to control the laser beam. The possible pulse control sources are:

- *Lens Centering*
- *Mirror Align Warm-up*
- *Mirror Align Flash*
- *Manual Control*
- *Cut (1, 2, 3 or 4)*
- *Pierce (1, 2, 3 or 4)*
- *Rapid Pierce (1, 2, 3 or 4)*
- *Aux Cut (1, 2, 3 or 4)*
- *Beam Off*

***Keyboard Shortcut:{ Alt, L, S} sequence***

## AUTO START-UP

**Laser | Auto Start-up** opens the *Laser Start-up* window, which will allow the user to enable or disable automatic resonator startup on a daily basis. Start-up time is entered separately for each day of the week. *Automatic Startup* can be enabled or disabled for each day of the week, or disabled entirely.




To use the *Automatic Startup* feature, keep the machine main power disconnect on, the chiller power disconnect on and set the chiller System switch to "REMOTE". In addition, the Laser System CNC/HMI program must be running at the programmed startup time, and the 'Automatic Operation Enabled' box must be checked. Select the "Start Now" button to immediately initiate a resonator startup.

When the programmed startup time arrives, the control will command the laser resonator Mains to turn on. This allows the chiller to circulate water at the required temperature before operation, and initiates the resonator startup sequence. If the control senses the chiller has not turned on, the automatic startup is aborted. The **Automatic Startup** function is complete when the resonator startup sequence reaches the *Ready* state.

*Note: The **Automatic Laser Start-up** function will not automatically turn on resonator high voltage. The "High Voltage" button on the Operator Console Control Bar must be manually pressed to activate high voltage and complete the resonator startup sequence.*

***Keyboard Shortcut:{ Alt, L, A} sequence***

## AUTO SHUT-DOWN

Using the **Automatic Shut-down** function, the Laser System can be programmed to automatically enter an idle state when the currently active cutting program terminates. In this state, the machine's servo drives are off (inhibited) and the resonator high voltage is off, but the control will remain active; The Laser System is now ready for an extended period of inactivity.

**! WARNING !**

**CINCINNATI INCORPORATED DOES NOT RECOMMEND COMPLETELY UNATTENDED OPERATION OF A LASER CUTTING SYSTEM. ALTHOUGH THE OPERATOR NEED NOT BE PRESENT WHILE THE LASER RUNS IN "UNATTENDED" MODE, SOME QUALIFIED PERSONNEL (MAINTENANCE, SECURITY, ETC.) MUST BE AVAILABLE TO DEAL WITH EMERGENCY SITUATIONS.**

### Arming the Automatic Laser Shut-down Function

To arm the **Automatic Laser Shut-down** function, select the **Auto Shut-down** command from the *Laser* menu. When the **Automatic Laser Shut-down** function is armed, a check mark will appear next to the menu command.

*Note: The **Auto Shut-down** command is only enabled while program execution is in progress. When the Laser System is not executing a cutting program, the command will appear grayed, indicating the **Automatic Laser Shut-down** function is disabled.*

### Automatic Shut-down Initiation

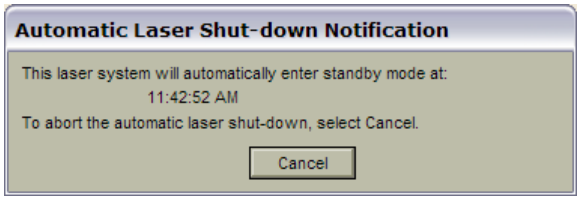
Once the **Automatic Laser Shut-down** function is armed, normal operation resumes until the active cutting program terminates. Program termination for any of the following reasons will trigger automatic shut-down:

- Normal program completion, for example: **M30**, **M02**, **M00**, or **M01** was encountered, presumably at the end of the program.
- Premature program termination caused by any system alarm or fault condition.
- Premature program termination caused by operator intervention such as pressing the "CYCLE STOP" or "EMERGENCY STOP" button.
- If program execution is suspended for longer than five minutes (for example: **M50/Switch Pallets** code is encountered and the Pallets "NOT READY" pushbutton is never pressed) the program will automatically terminate. In this case, behavior will be similar to normal program termination; no system alarms will be generated.
- The *Automatic Laser Shut-down* sequence begins immediately after the active program terminates.

### Automatic Shut-down Sequence

Whenever an active program is terminated while the **Automatic Laser Shut-down** function is armed, the control will execute an automatic shut-down sequence, performing the following tasks in the order listed:

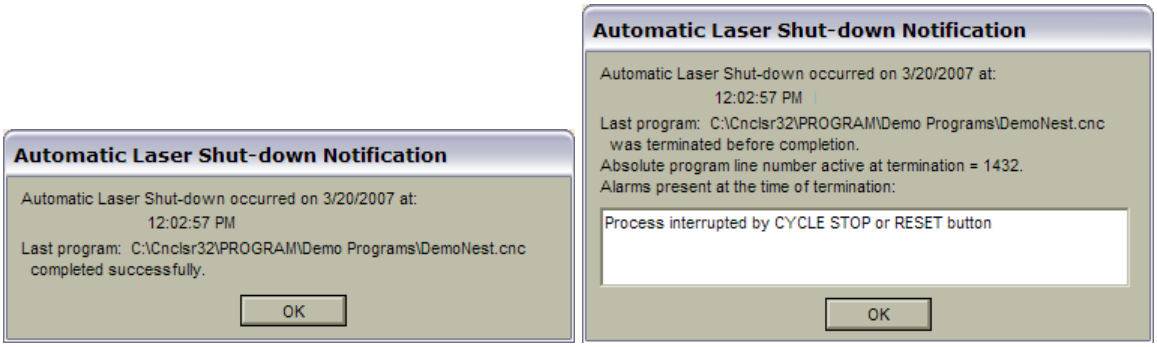
1. The *Shut-down Notification* window will pop up, notifying that automatic shut-down will occur at the indicated time. This message will be displayed for five minutes, during which time the user can abort the automatic shut-down if desired. Selecting "Cancel" during this five minute period will disarm the **Automatic Laser Shut-down** function and the control will resume normal operation.


Automatic Laser Shut-down Notification
This laser system will automatically enter standby mode at:
11:42:52 AM
To abort the automatic laser shut-down, select Cancel.
Cancel

2. If the shut-down is not aborted within five minutes, the servo drives, Mains, and resonator high voltage will be turned off.

3. The message displayed in the *Shut-down Notification* window will change, notifying that shutdown was completed and the time of actual shutdown (see examples). At this time, the notification window will also display:

   - The name of the program running when Automatic Shut-down occurred.
   - The completion status of the last program: "completed successfully" or "terminated before completion".
   - If the program completion status is "terminated before completion", a list of all system and laser alarms present at the time of program termination will be displayed. The absolute line number identifying which program block was active when termination occurred will also be displayed. This line number can later be used with the **Program Restart** function to restart the program at a specified line.

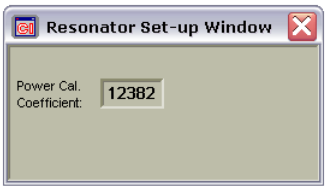

Automatic Laser Shut-down Notification
Automatic Laser Shut-down occurred on 3/20/2007 at:
12:02:57 PM
Last program: C:\Cnclsr32\PROGRAM\Demo Programs\DemoNest.cnc
completed successfully.
OK
Automatic Laser Shut-down Notification
Automatic Laser Shut-down occurred on 3/20/2007 at:
12:02:57 PM
Last program: C:\Cnclsr32\PROGRAM\Demo Programs\DemoNest.cnc
was terminated before completion.
Absolute program line number active at termination = 1432.
Alarms present at the time of termination:
Process interrupted by CYCLE STOP or RESET button
OK

*Note: Absolute line number identifies the sequential position of a row or block of code in a program file. This number is always relative to the first line in the program and is not necessarily the same as the optional program line number "Nxxxxx". When an NC Program Window is the active window, the absolute line number of the line containing the cursor is displayed in the Main Status Bar.*

***Keyboard Shortcut:*** ***{ Alt, L, H} sequence***

## RESONATOR SET-UP WINDOW

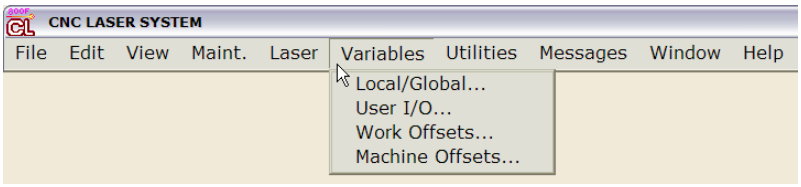
The *Resonator Set-up Window*, displays the *Power Calibration Coefficient* for the laser resonator. This read-only value can be used in resonator setup or diagnostic procedures.




***Keyboard Shortcut:*** ***{ Alt, L, R} sequence***

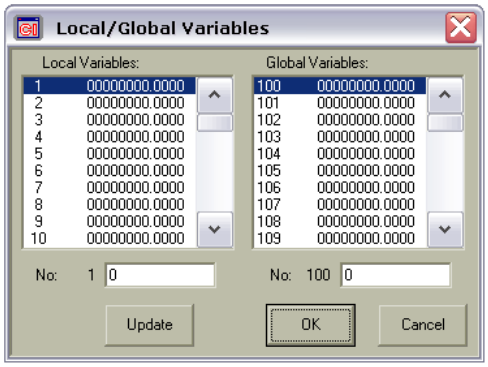
# VARIABLES MENU

The *Variables* menu commands provide access to windows that display variables and offset information used by NC cutting programs.




## LOCAL/GLOBAL VARIABLES

**Variables | Local/Global** allows the user to view and edit the local (#1 to #99) and global (#100 to #999) variables. All variables can be edited.



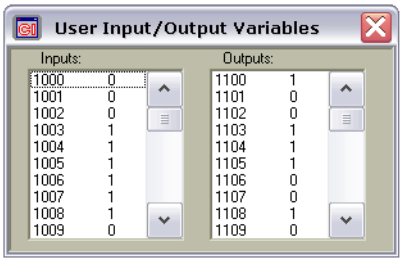

**CAUTION**

**Editing variables while a program is running can cause improper execution and is strongly discouraged.**

***Keyboard Shortcut: { Alt, A, L} sequence***

## USER I/O VARIABLES

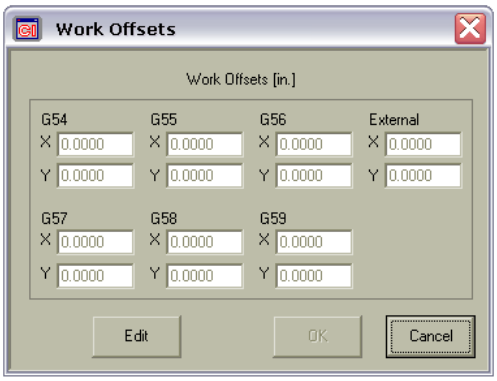
**Variables | User I/O** displays user I/O variables. The user I/O variables cannot be edited; they can only be changed through program execution.




***Keyboard Shortcut: { Alt, A, U} sequence***

## WORK OFFSETS

**Variables | Work Offsets** allows the user to view or edit the offsets for work coordinate systems 1 through 6 (**G54** to **G59**) and the External Work Offsets. The user can edit work coordinate offsets only when a program is not executing. (See **G54…G59 Work Coordinate System Selection** in the **LASER NC PROGRAMMING** topic.)



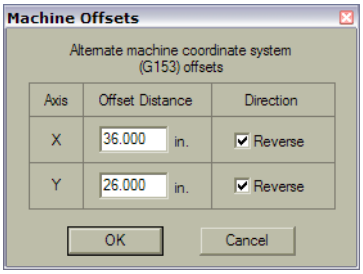

***Keyboard Shortcut: { Alt, A, W} sequence***

## MACHINE OFFSETS

**Variables | Work Offsets** allows the user to view or edit the offsets for the alternate machine coordinate system. The user can edit machine coordinate offsets only when a program is not executing.

*Offset Distance:* These settings specify the distance from the Axes Home position to the **X**0 or Y0 position in the alternate machine coordinate system.

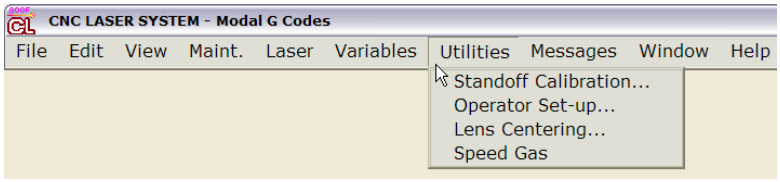
*Direction:* The 'Reverse' check box specifies if the direction of the alternate machine coordinate system axis is opposite the standard machine axis direction. When the X or Y Offset Distance is not zero, the *Axes Positions* window and the *All Positions* window include the nozzle position in the alternate machine coordinate system.




***Keyboard Shortcut: { Alt, A, M} sequence***

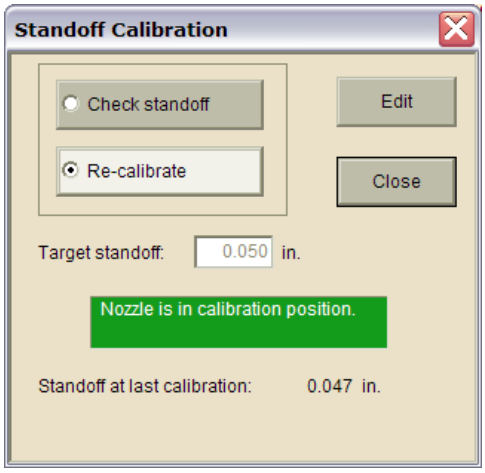
# UTILITIES MENU

Operators can use the *Utilities* menu to access setup and calibration functions that are performed on a regular basis.




## STANDOFF CALIBRATION

**Utilities | Standoff Calibration** opens the *Standoff Calibration* window and puts the Laser System in Standoff Calibration mode. This is a "Special Operating Mode", used to check or calibrate the noncontact head height sensing system.




Periodic use of the automated **Standoff Calibration** function will ensure an accurate standoff between the nozzle tip and the workpiece. When the "CYCLE START" button is pressed, the cutting head moves down until the nozzle tip touches the workpiece and then retracts to the desired standoff using the Z-axis motor encoder. This provides an accurate and repeatable method for independently verifying the standoff accuracy of the capacitive or "Noncontact" sensor. See **NOZZLE STANDOFF CALIBRATION** in the **CALIBRATION AND ADJUSTMENTS** section for more about standoff calibration.

***Toolbar Shortcut:***

***Keyboard Shortcut: { Alt, U, T} sequence***

## OPERATOR SETUP

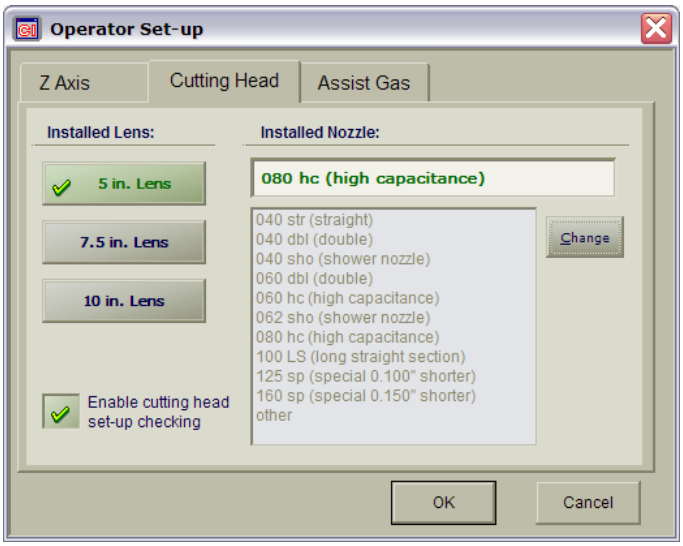
**Utilities | Operator Set-up** opens the *Operator Set-up* window. This window is used to specify how the cutting head and assist gas ports are currently set up (required for the **Machine Setup Checking** function), and to access various Z-axis setup parameters.

The *Operator Set-up* window can be opened at any time, and all of the set-up parameters can be edited by the operator user. Note however, that changing set-up parameters while a part program is executing is not permitted. The *Operator Set-up* window consists of three separate pages or tabs:

- Cutting Head Setup
- Assist Gas Setup
- Z-axis Setup

## Cutting Head Setup

The Cutting Head page of the *Operator Set-up* window is used to enable/disable cutting head set-up checking and to specify how the cutting head is currently set up.



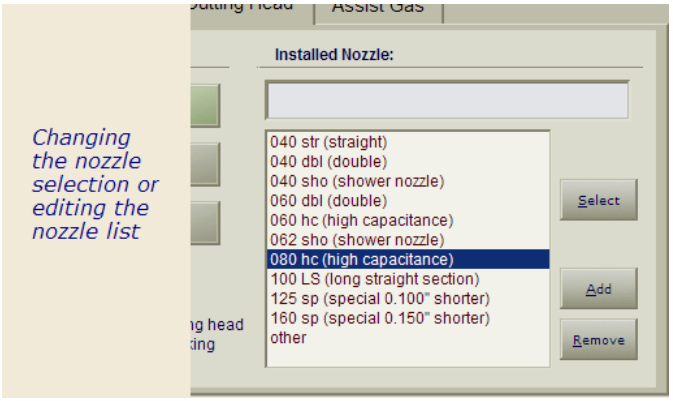

*Enable cutting head set-up checking*: Check this box to enable cutting head setup checking. If cutting head set-up checking is enabled, then the **Machine Setup Checking** function will issue a machine set-up error if the current focusing lens or nozzle tip is different than that which is specified by a Process Parameter file used by the active part program. If cutting head setup checking is not enabled, then the **Machine Setup Checking** function will not check for focusing lens or nozzle tip conflicts.

*Installed Lens*: This set of buttons indicates which focusing lens is currently installed in the cutting head. If cutting head setup checking is enabled, the operator is required to click the appropriate lens button whenever the lens is physically changed in the cutting head.

*Installed Nozzle*: This field indicates which nozzle tip is currently installed in the cutting head. If cutting head setup checking is enabled, the operator is required to select the matching item from the nozzle list whenever the nozzle is physically changed in the cutting head.

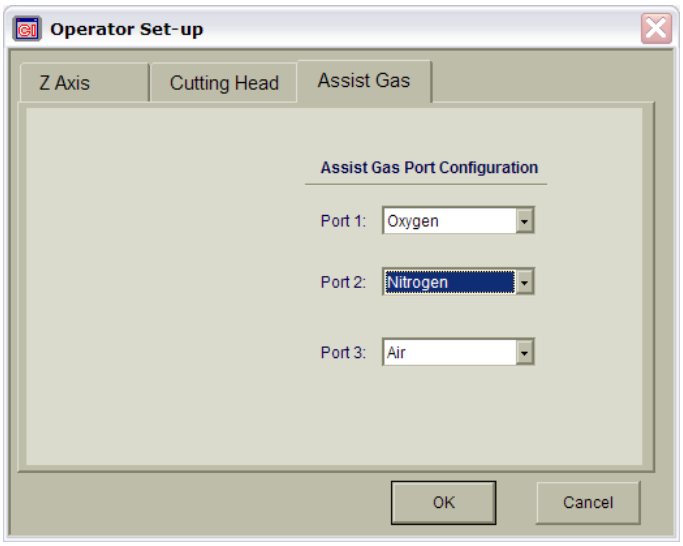
*Nozzle List*: The list of available nozzle tips is just below the "Installed Nozzle" field. The operator can edit the nozzle list if needed. This manual shows the factory-supplied list. To change the installed nozzle setting or edit the nozzle list, first click the "Change" button. Three new buttons, "Select", "Add" and "Remove" will then appear (see figure).

- To change the installed nozzle setting, select the desired nozzle from the list, then click "Select".
- To edit the nozzle list, click "Add" to insert a new nozzle into the list or "Remove" to delete a nozzle from the list. Clicking a selected nozzle list item a second time will activate a single line editor, which can be used to modify that nozzle description. A nozzle description can be any combination of alpha and numeric characters up to 32 characters in length.



## Assist Gas Setup

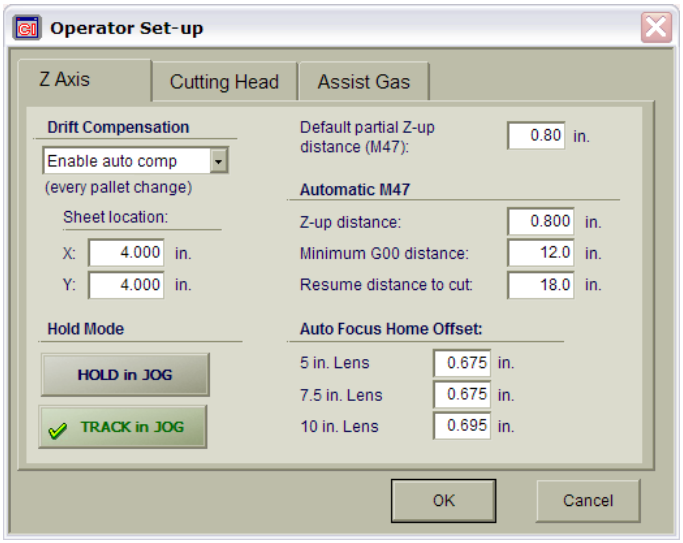
The *Operator Set-up* window Assist Gas page is used to specify how the HMI control displays and manages assist gasses.




*Assist Gas Port Configuration*: This is the assist gas port map, which is where each assist gas port is associated with a specific type of gas. The assist gas port map consists of a drop-down list for each assist gas port. Use this map to specify which assist gas is currently connected to each port.

## Z-axis Setup

The Z-axis Setup page of the *Operator Set-up* window contains configuration settings related to the Z-axis that the operator may want to change as part of normal machine operation.




## Drift Compensation

Changes in the cutting environment can cause standoff accuracy to degrade over time. **Drift Compensation** is a function that re-calibrates the standoff after every pallet exchange, to correct for standoff drift. This function can operate in one of three modes:

*Off:* The **Drift Compensation** function is disabled

*Enable Auto Comp:* In this mode, the CNC control will automatically check nozzle standoff after each pallet exchange. The cutting head moves to the machine coordinates specified in the "Sheet Location" fields and performs a Standoff Calibration move, as described in **NOZZLE STANDOFF CALIBRATION** in the **CALIBRATION AND ADJUSTMENTS** section. Any error detected becomes the new compensation amount or "offset". For example, if the nozzle is 0.002 inch (0.05 mm) lower than required for a 0.050 inch (1.27 mm) standoff, the control will compensate by commanding 0.052 inch (1.32 mm) standoff. The control uses the new offset value until the load pallet moves into the cut position.

*Lower Pallet Offset:* The CNC control will automatically compensate the standoff when cutting on the lower pallet, by the amount specified in the "Lower pallet offset" field.

*Note: To determine if lower pallet compensation is necessary, first set Drift Compensation to "OFF", and then check standoff calibration on both pallets, beginning with the upper pallet. If the procedure indicates any difference in calibration between the two pallets, select "Lower pallet offset" and enter the difference in the field. Recheck calibration on both pallets and restore Drift Compensation to "ON" if desired.*

### Automatic M47

This feature helps to protect the cutting head during high-speed moves. The control "looks ahead" to see if the next X/Y rapid move is longer than the distance specified in the "Minimum G00 distance" field. If the next rapid move is too long, the control raises the cutting head as the X- and Y-axes begin to move. When the X- and Y-axes approach the end of the move, the cutting head moves down, to be in position for the next cut. (The Z-move is trapezoidal)

*Z-up Distance:* The distance that the head retracts during Automatic Partial Z-up moves.

*Minimum G00 Distance:* If the length of the X/Y rapid move exceeds this value, the head will automatically retract. Otherwise, the head will track the material during the rapid move.

*Note: To disable Automatic Partial Z-up moves, enter a large **Minimum G00 Distance** (for example, 999 inches or 25000 mm).*

*Resume Distance to Cut:* When the distance remaining in a long rapid move is equal to or less than this value, the control commands the cutting head to resume tracking.

*Default Partial Z-up Distance (M47):* The Z-axis raises the nozzle this distance when the program commands **M47**. The default value of 0.800 inches (20 mm) is sufficient to clear material clamps. However, some applications may require higher distances in order to avoid tipped parts or slugs.

*Note: The program can override this default by specifying a distance with the 'P' argument in the **M47** block. Refer to **M Codes** in the **LASER NC PROGRAMMING** section for more on **M47**.*

### Hold Mode

*HOLD in JOG:* When this button is selected, the cutting nozzle remains locked at its current vertical position during all jog moves, i.e., the cutting head does not track the material height.

*Note: X-axis and Y-axis jog motion will not be allowed unless the nozzle is at least 2.0 inches (50.8 mm) above the top of material.*

*TRACK in JOG:* When this button is selected, the cutting nozzle position will change during jog moves as necessary to maintain the last commanded standoff. This is the default Jog mode.

### Auto Focus Home Offset

For Laser Systems equipped with the *Auto Focus* cutting head system, the *Z-axis Setup* window includes parameters for Auto Focus home offset. See the Auto Focus Cutting Head section for a description of these parameters.

***Toolbar Shortcut:***

***Keyboard Shortcut:*** ***{ Alt, U, O} sequence***

## LENS CENTERING

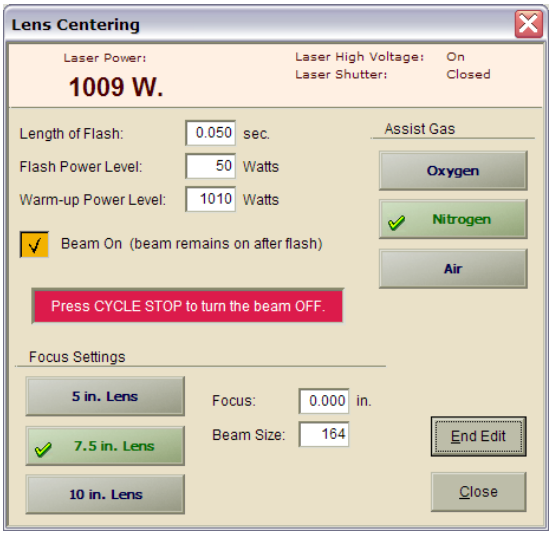
The **Utilities | Lens Centering** command opens the *Lens Centering* window and puts the laser in the "Special Operating Mode" for executing "tape shots". In Lens Centering mode, the laser beam is activated directly by the "CYCLE START" button on the Operator Control Station front panel, or the "SHUTTER FLASH" button on the Remote Station. See **CENTERING THE FOCUS LENS** in the **CALIBRATION AND ADJUSTMENTS** section, for more on the lens centering procedure.

**! ! DANGER ! !**

**KEEP HANDS AWAY FROM BEAM PATH DURING A LASER SHOT.**

*Note: When the **Lens Centering** window is open, the Laser System is in a special diagnostic mode used only for machine setup/maintenance. Normal machine operation such as program execution is disabled while the Lens Centering mode is active.*

From the *Lens Centering* window, the laser can be configured to execute "tape shots" by setting the power level, assist gas port and beam flash time. To change tape shot parameters, select the "Edit" button, make the desired changes, and then select "End Edit".




*Length of Flash:* Specifies how long the beam will be on with the shutter open during a tape shot.

*Flash power level:* The power level at which the beam will be "flashed" during a tape shot.

*Warm-up power level:* Specifies the power level that the laser will maintain before and after the tape shot if "Beam On" is checked. If "Beam On" is not checked, the Warm-up power level is not displayed.

*Beam On:* This check box configures the laser to keep the beam on before and after the shutter flash. To turn on the laser beam, press "CYCLE START". In this mode, the laser will maintain the Warm-up power level while the shutter is closed. This mode is designed to simulate the thermal condition of the internal laser optics during normal operation. To turn off the laser beam, press "CYCLE STOP".

*Assist Gas:* Use these buttons to specify which assist gas to use during the tape shot.

*Focus Settings:* Use these settings to specify the beam size, focusing lens and focus position to be used for a tape shot.

*Focus:* This is the focus position (the location of the focal point relative to the nozzle tip) during the beam flash. For tape shots, the focus position is normally set to 0.000 in. (The lens "home" position.). See the **AUTO FOCUS CUTTING HEAD** section for more about beam focus.

*Beam Size:* This setting affects the diameter of the laser beam during the beam flash. Decreasing the *Beam size* setting will result in a smaller beam diameter while increasing the setting will produce a larger beam diameter

***Toolbar Shortcut:***

***Keyboard Shortcut:*** ***{ Alt, U, L} sequence***

### SPEED GAS

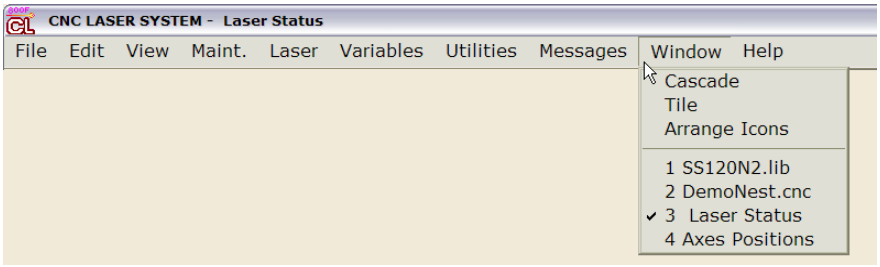
The **Utilities | Speed Gas** submenu shows the current status of the **Speed Gas** function; a check mark indicates the function is on. Selecting this menu item will toggle Speed Gas on or off.

When Speed Gas is on, assist gas continues to flow through the nozzle after a program commands **Beam OFF** (**M35**). This function eliminates the delay to turn on assist gas and stabilize pressure when the program commands the next **Beam ON** (**G84** or **G85**) command. This function can reduce cycle time when a program commands many small cutting paths close together.

***Keyboard Shortcut:*** ***{ Alt, U, S} sequence***

## WINDOW MENU

The *Window* menu commands can be used for arranging and accessing windows when the *Laser System Display* window has multiple windows open at the same time.




### CASCADE

**Window | Cascade** arranges multiple opened windows in an overlapped fashion.

***Keyboard Shortcut:*** ***{ Alt, W, C} sequence***

### TILE

**Window | Tile** arranges multiple opened windows in a non-overlapped fashion.

***Keyboard Shortcut:*** ***{ Alt, W, T} sequence***

### ARRANGE ICONS

**Window** | **Arrange Icons** arranges icons of minimized windows at the bottom of the *Laser System Display* window's Application Workspace. If an open window is at the bottom of the Application Workspace, then some or all of the icons may not be visible if they are underneath the window.

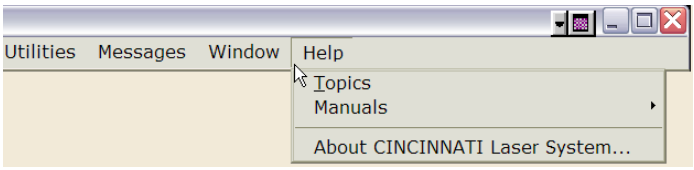
***Keyboard Shortcut:*** ***{ Alt, W, A} sequence***

### WINDOW 1, 2 ...

A list of currently open windows is displayed at the bottom of the *Window* menu. A check mark appears in front of the name of the active window. Choose a window from this list to make it the active window.

## HELP MENU

Use the *Help* menu commands to get information about the CINCINNATI Laser System.



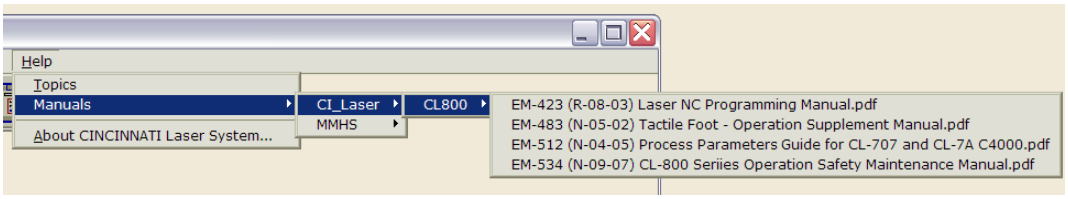

### TOPICS

The **Help | Topics** command displays starts the **HTML Help** application, displaying this Laser System Help document on the Operator Control Station monitor.

***Keyboard Shortcut:*** ***F1 or { Alt, H, T} sequence***

### MANUALS

The **Help | Manuals** command scans the folder "D:\Manuals" and opens in fly-out submenus displaying links to all files and subfolders it encounters. This provides fast, easy access to documents provided by CINCINNATI for Laser System users.

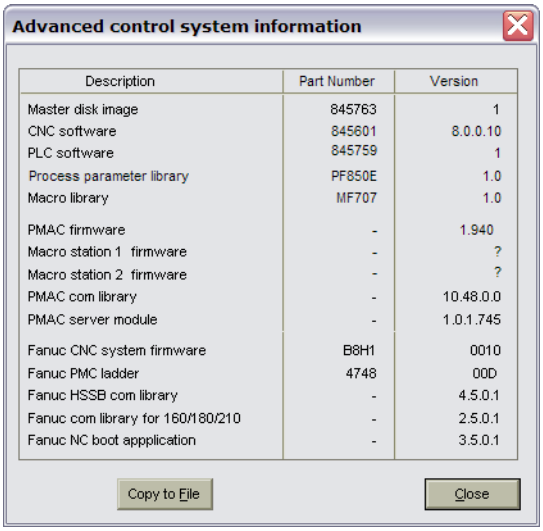


## ABOUT CINCINNATI LASER SYSTEM

The **Help | About CINCINNATI Laser System** command opens the *About CNC Laser Control* window, which displays the Laser System's machine model and serial number, and control software version information.

About CNC Laser Control
CINCINNATI
CINCINNATI INCORPORATED
Cincinnati CL-850 F Laser System
Machine Serial Number: 66830
CNC Software Part Number: 845601 Version: 8.0.0.10
PLC Software Part Number: 845759 Version: 3
Copyright © 1996 - 2006 Cincinnati Incorporated
Advanced Information
OK

*Advanced Information:* This button opens the *Advanced Control System Information* window, which displays additional control software and driver version information.


Advanced control system information
Description | Part Number | Version
Master disk image | 845763 | 1
CNC software | 845601 | 8.0.0.10
PLC software | 845759 | 1
Process parameter library | PF850E | 1.0
Macro library | MF707 | 1.0
PMAC firmware | - | 1.940
Macro station 1 firmware | - | ?
Macro station 2 firmware | - | ?
PMAC com library | - | 10.48.0.0
PMAC server module | - | 1.0.1.745
Fanuc CNC system firmware | B8H1 | 0010
Fanuc PMC ladder | 4748 | 00D
Fanuc HSSB com library | - | 4.5.0.1
Fanuc com library for 160/180/210 | - | 2.5.0.1
Fanuc NC boot appplication | - | 3.5.0.1
Copy to File
Close

*Copy To File:* The user can use this button to write the advanced control system information to a text format file on the laser control PC hard drive. The text file will be named *"SysFileInfo.txt"*, in the folder "*C:\CncLsr32*".

***Keyboard Shortcut:*** ***{ Alt, H, A} sequence***

# CUTTING PROCESS PARAMETERS

The purpose of this section is to briefly describe the parameters associated with the laser cutting process and explain how to adjust those parameters on the Laser System. This section does not include recommended cutting parameters or guidelines for improving the cutting process with particular materials. For that information, refer to the **Process Parameters and Diagnostic Guide**.

## INTRODUCTION

Laser cutting is a thermal cutting process that uses the energy of a focused laser beam in conjunction with a controlled gas jet. The focused laser beam is directed at a very small portion of the workpiece and melts/vaporizes the workpiece material in that area. The controlled gas jet (assist gas) is directed at the same area and removes the molten/vaporized material. In some cases, the assist gas also adds thermal energy to the process from an oxidation reaction with the molten material.

All of the parameters associated with the laser cutting process fall into one of three categories: control of laser power, control of the focusing characteristics of the laser beam, or control of the assist gas. The laser power can be continuous wave (CW) or pulsed. The basic parameters associated with controlling laser power are the power level, the pulse frequency, and the duty cycle.

The focusing characteristics of the laser beam are controlled by the choice of lens focal length and by adjustment of the focal point. The lens focal length is the distance from the center of the lens to the point where the beam diameter is smallest (the focal point). The position of the focal point relative to the workpiece is a critical parameter in the laser cutting process.

The basic parameters associated with assist gas are the gas type and pressure. In addition, the assist gas flow is controlled by the choice of nozzle tip and by the nozzle standoff (the distance from the nozzle tip to the workpiece).

The laser cutting process consists of two basic steps – pierce and cut. The pierce process produces a starting hole in the workpiece. Usually the cutting head is stationary during the pierce process. Moving the cutting head with laser power on and with assist gas flowing produces the laser cut. For most materials, different parameters are used for the pierce and the cut.

For thin metals, the pierce process is very simple and requires only a basic set of parameters. For thicker metals, more specialized techniques are used for the pierce process. These techniques include ramped pierce and Rapid Pierce. The parameters associated with these techniques are described in the following sections.

For many materials, especially thick metals, special techniques are also incorporated to improve the cutting process. Most of these techniques are associated with starting the cut, cutting sharp corners, or cutting small features. The techniques provided to improve the start-of-cut include precut dwell, power burst and optional assist gas pressure. The techniques provided to improve the cutting of sharp corners and small features include Dynamic Power Control. The parameter adjustments for these techniques are described in the following sections.

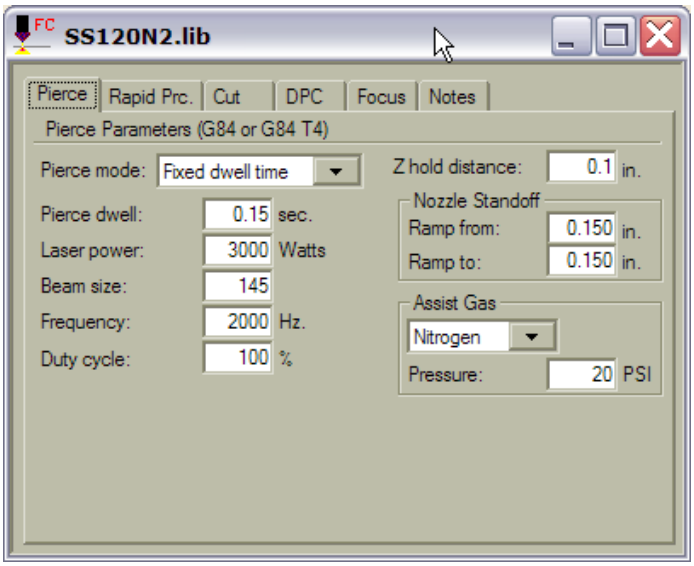
## PROCESS PARAMETER WINDOW

Cutting Process Parameters are viewed/edited through a *Process Parameter* window. Process Parameters are organized into several categories. Each group of parameters is displayed on a separate page in the *Process Parameter* window. To display a page, select the corresponding tab at the top of the *Process Parameter* window. The names of the Process Parameter groups and their corresponding tabs are:

- *Pierce* Pierce parameters
- *Rapid Prc.* Rapid Pierce parameters
- *Ramped Prc.* Ramped Pierce power parameters
- *Cut* Cut parameters
- *DPC* Dynamic Power Control parameters
- *Focus* Auto focus and Dynamic Assist Gas parameters
- *Notes* Process Parameter notes

## PIERCE PARAMETERS

The Pierce Parameters page contains the following parameters – *Z hold distance*, *Pierce mode*, *Pierce dwell*, *Laser power*, *Frequency*, *Duty cycle*, *Beam size*, *Nozzle Standoff*, *Assist Gas Port*, and *Assist Gas Pressure*. These parameters specify how the Laser System responds to **G84** (normal pierce, then cut sequence) or **G84 T4** (pierce-only sequence) codes in a part program. See **G84 AND G85 START CUTTING SEQUENCE** in the **LASER NC PROGRAMMING** topic for more information on **G84**.




*Z hold distance:* This parameter specifies how far the noncontact head must move away from the pierce (in the x or y direction) before the Z-axis control will respond to material height changes. The Z-axis control always maintains a fixed position during the pierce sequence The *Z hold distance* parameter extends that holding function to help start the cutting process. This technique can improve cut quality if material slag builds up on the surface during a pierce.

*Pierce mode:* This setting specifies the type of pierce, if any, to be used and how the Pierce Parameters page is displayed. The following pierce modes available:

- *No Pierce:* **G84** execution commences with the cut cycle. No pierce sequence is executed. In this mode, the Pierce Parameters page is blank, except for the *Z Hold Distance* and *Pierce mode* settings. In this mode, **G84** functions the same as **G85**.
- *Fixed dwell time:* **G84** initiates a pierce sequence that will terminate after *Pierce dwell* seconds elapses. This is followed by the cut cycle.
- *Ramped pierce*: **G84** initiates a ramped-power pierce sequence based on the Ramped Pierce Parameters page settings, followed by the cut cycle.
- *Adaptive pierce*: Adaptive pierce mode is available only for lasers equipped with the optional Process Monitor system described later in this section.

*Pierce dwell:* Specifies the duration of the pierce sequence. If "Ramped Pierce Standoff" is used, the nozzle will move from the initial pierce standoff (*Ramp from* value) to the final pierce standoff (*Ramp to* value) in *Pierce dwell* seconds. This also applies to the **Ramped Pierce Focus** function (see **BEAM FOCUS POSITION**).

*Note: In Ramped pierce mode, the "Pierce dwell" field cannot be edited, and displays the sum of the ramp step times on the Ramped Pierce Parameters page.*

*Laser power:* Specifies the peak laser power in each pulse during the pierce sequence.

*Frequency and Duty cycle:* These parameters determine the frequency and duty cycle of the power waveform when pulsed laser power is used. To use continuous wave (not pulsed) power for piercing, set the duty cycle to 100%. In pulsed laser power mode, the pulse on time must be at least as long as the minimum pulse time attainable by the resonator. If a duty cycle value results in a pulse on time that is too small, the user will be prompted to enter a valid duty cycle before saving the file. In this case, the prompt message will display the range of duty cycle values the will produce an acceptable pulse on time for the given *Frequency* setting.

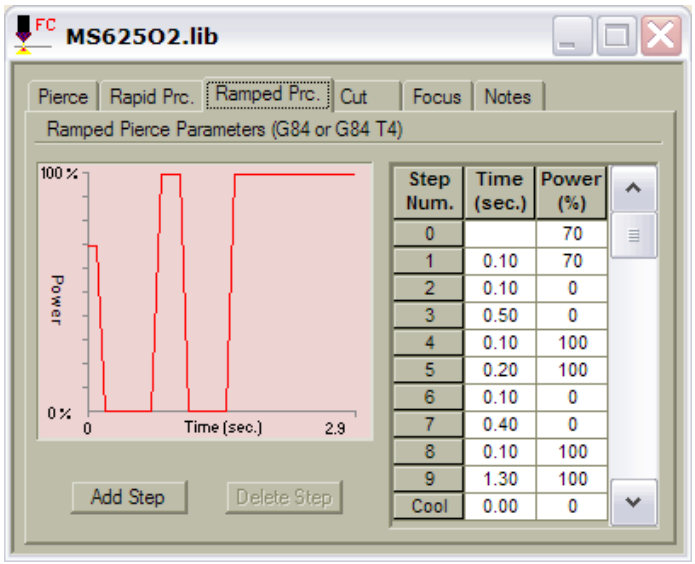
*Beam size:* Changes the diameter of the laser beam produced during the pierce sequence. Decreasing the Beam size setting will result in a smaller beam diameter while increasing the setting will produce a larger beam diameter.

*Nozzle Standoff:* The distance from the bottom of the nozzle tip to the workpiece surface. The standoff can be configured to change (ramp) during the pierce sequence. A ramped pierce standoff will result when the *Ramp from* parameter has a different value than *Ramp to*. In this case, the pierce sequence will start with the nozzle at the *Ramp from* standoff distance and end at the *Ramp to* distance.

*Assist Gas:* Use these settings to specify which assist gas and what pressure to use for the pierce sequence.

## RAMPED PIERCE PARAMETERS

The Ramped Pierce Parameters page is used to configure a ramped power pierce, which is a pulsed pierce in which the laser power duty cycle changes with time. This page contains a table that allows the programming of a power ramp pattern consisting of up to 20 steps. Each step defines a power value and time. The laser power changes linearly from the previous value to the new value during the time specified for each step, creating a user-programmable power versus time pierce function. The last step provides a *Cool* time. The laser beam is turned off during this time to allow the work material to cool before starting the cut process. A graph shows the power versus time profile. To specify ramped pierce in a cutting program, use **G84** or **G84 T**4 when the pierce mode of the active Process Parameter file is set to "Ramped power".



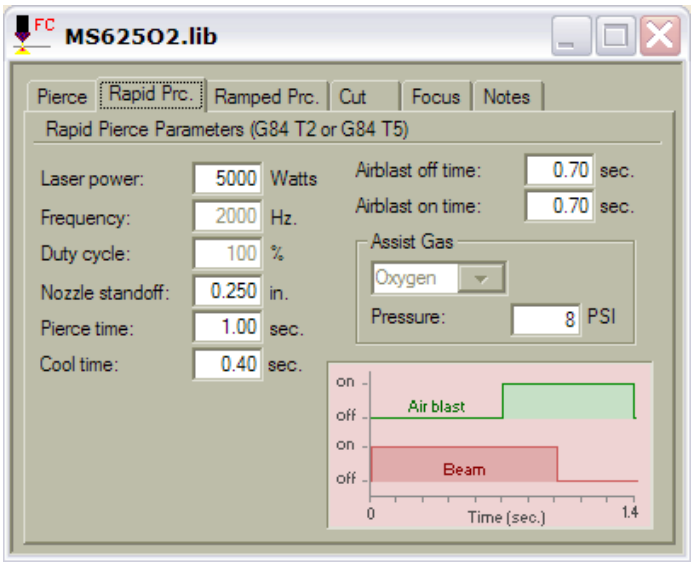

The average power level is controlled by varying the duty cycle, not the peak power in each pulse. The peak power of each pulse is held constant at the value established by the *Laser power* setting on the Pierce Parameters page. The actual duty cycle at any time in the pierce process is equal to the product of pierce duty cycle and the ramp power percentage at that time. For example, if the pierce duty cycle is 80% and the power setting is 60% for ramp set 2, the duty cycle at the end of ramp step 2 is 48%.

*Note: The Ramped Prc. tab of the Ramped Pierce Parameters page is visible only if Pierce mode on the Pierce Parameters page is set to "Ramped power".*

## RAPID PIERCE PARAMETERS

**Rapid Pierce** is an optional feature that reduces the pierce time for thick mild steel. The pierce hole produced by a Rapid Pierce is shaped like a cone and considerably larger than the pierce hole produced by the standard (pulsed) pierce. The pierce process also creates a small mound of molten material surrounding the top of the pierce hole. This pierce mound is removed with a short blast of compressed air immediately after the laser beam pierces the material. To specify *Rapid Pierce* in a cutting program, use **G84 T**2 (Rapid Pierce, then cut sequence) or **G84 T**5 (Rapid Pierce-only sequence).

If the *Rapid Pierce* option is enabled ("Rapid Pierce" is checked on the Options page of the *Machine Configuration* window), then each Process Parameter window includes a page specifying *Rapid Pierce* parameters. The parameters associated with Rapid Pierce are *Laser power*, *Nozzle standoff*, *Pierce time*, *Cool time*, *Airblast off time*, *Airblast on time*, and *Assist Gas Pressure*. The assist gas type setting on this page cannot be edited since the **Rapid Pierce** function uses the same assist gas as a normal pierce.




*Laser power:* Specifies the level of continuous laser power during the **Rapid Pierce** cycle. Since *Rapid Pierce* uses a continuous wave laser beam, *Frequency* and *Duty cycle* are displayed here for reference, but cannot be edited.

*Nozzle standoff:* This is the distance from the bottom of the nozzle tip to the workpiece surface.

*Pierce time:* This is the duration of the **Rapid Pierce** process.

*Cool time:* Specifies a time following the pierce time during which the laser beam is turned off. The cool time allows the work material to cool before starting the cut process.

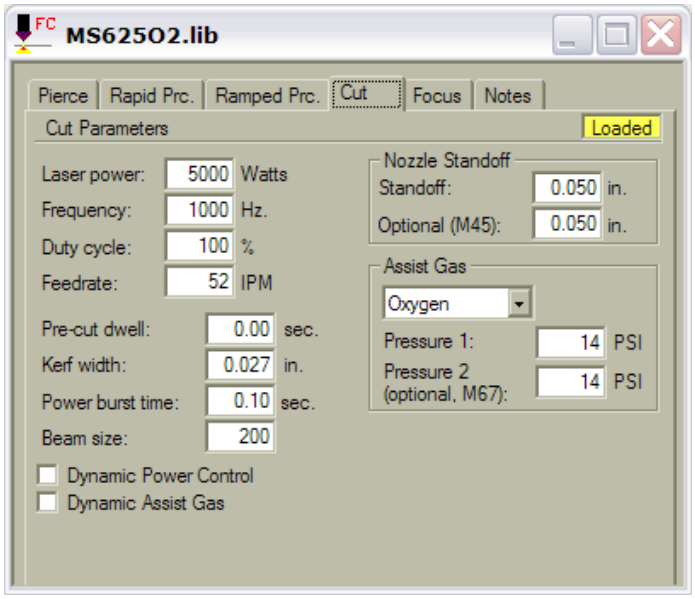
*Airblast off time*

*Airblast on time:* These two parameters control the opening of the airblast solenoid valve. *Airblast off time* is the time from the beginning of the pierce to the activation of the airblast valve. *Airblast on time* specifies the length of time the airblast valve is activated. The Rapid Pierce Parameters page contains a graph that shows the timing of the airblast relative to the laser beam.

*Note: The sum of **Airblast off time** plus **Airblast on time** must be less than or equal to the sum of **Pierce time** plus **Cool time**. If this requirement is not satisfied when any of the time values are edited, the new time value will be rejected and the user will be prompted to adjust **Cool time** accordingly.*

## CUT PARAMETERS

The Cut Parameters page contains the parameters that define how the cutting sequence is executed following **G84** or **G85**: *Laser power*, *Frequency*, *Duty cycle*, *Feedrate*, *Pre-cut dwell*, *Kerf width*, *Power burst time, Beam Size, Nozzle Standoff*, *Assist Gas Port*, *Assist Gas Pressure 1*, and *Assist Gas Pressure 2 (Optional, **M67**)*. This page also contains check boxes to select "Dynamic Power Control" and "Dynamic Assist Gas".




*Laser power:* The peak laser power in each pulse during the cutting process. Note that the actual, average power produced by the laser is equal to the product of the *Laser power*, *Frequency*, and *Duty cycle* parameters.

*Frequency* and *Duty cycle:* These parameters determine the pulse characteristics of the pulsed laser power (when "Dynamic Power Control" (DPC) is not selected). When DPC is selected, the DPC parameters also control the frequency and duty cycle. Refer to the discussion of *Dynamic Power Control* parameters for more details. To use CW (continuous wave) mode for cutting, set the duty cycle to 100%.

*Feedrate:* This parameter should be the maximum cutting speed at which good edge quality can be achieved in normal operation. For some part features, the actual cutting speed might be reduced to maintain dimensional accuracy of the part or to improve the edge quality of a feature. Execution of the **G89** command can set a variable (default is variable #148) equal to the value of the cut feedrate parameter. This ability to make the feedrate parameter available to a program is configurable. See **PROCESS LIBRARY FEEDRATE** in the **GENERAL CONFIGURATION** topic for more on enabling the use of the cut feedrate parameter. Note that the **G89** command only sets the value of a variable; the X-axis and Y-axis cutting speed does not change until the program commands the feedrate with (for example) "**F#148**".

*Pre-cut dwell time:* Specifies a dwell at the beginning of the cut process before motion of the cutting head begins. At the end of the pierce process, all parameters are switched to cut parameters with the laser beam on. Motion does not begin until the pre-cut dwell time has elapsed.

*Kerf width:* The width of the kerf produced by the cutting process. The control uses this parameter for cutter compensation if the program commands **G41** or **G42**.

*Beam size:* This setting affects the diameter of the laser beam produced during the cutting process. Decreasing the beam size setting will result in a smaller beam diameter while increasing the setting will produce a larger beam diameter.

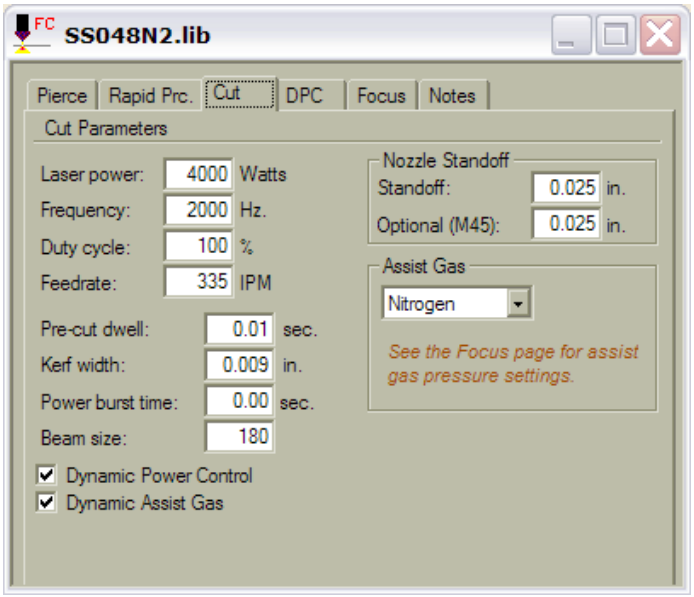
*Dynamic Power Control:* Enables the **Dynamic Power Control (DPC)** function. When this box is checked on the Cut Parameters page, the Process Parameter window includes *Dynamic Power Control* parameters on a separate page labeled "DPC".

*Power burst time:* The **Power Burst** function can be used to delay the start of **Dynamic Power Control** at the beginning of the cut sequence. *Power burst time* is the period after motion has begun that laser power is held at the values specified in the "Laser power" and "Duty cycle" fields. DPC is initiated as soon as the power burst time delay elapses. If the power burst time is set to zero, then the **DPC** function begins controlling laser power as soon as the cut sequence starts. If "Dynamic Power Control" is not selected, then *Power burst time* has no effect on laser power.

*Dynamic Assist Gas:* Enables the **Dynamic Assist Gas (DAG)** function. When this box is checked on the Cut Parameters page, the Focus Parameters page includes a section for *Dynamic Assist Gas* parameters.

*Nozzle Standoff:* Standoff is the commanded distance from the workpiece surface to the bottom of the nozzle tip used for normal cutting sequences. When a program commands **M45**, the nozzle standoff changes to the "Optional (**M45**)" distance.

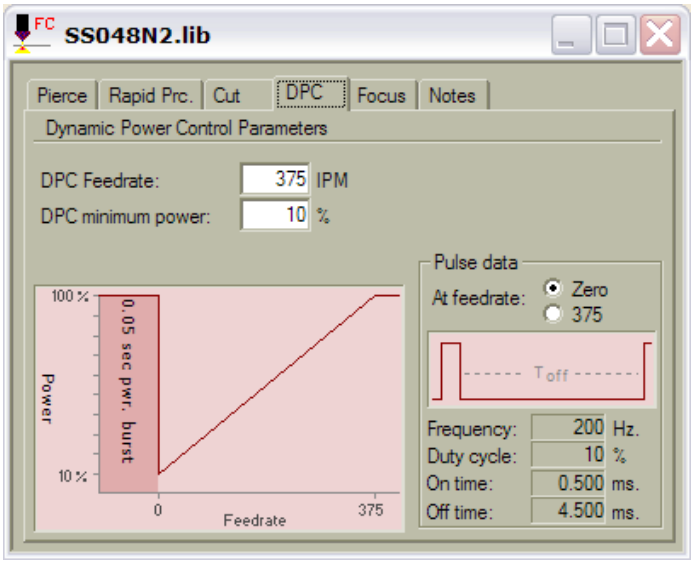
*Assist Gas:* Use these settings to specify which assist gas and what pressure to use for the cutting process. The assist gas settings specified on the Cut Parameters page will be activated as soon as the pierce sequence is complete. The assist gas pressure for the cut process will depend on the *Dynamic Assist Gas* setting. If DAG is off, the pressure will change to the value in the "Pressure 1" field, as shown in the figure above. If the program commands **M67**, the pressure will switch to the value in the "Pressure 2 (Optional, **M67**)" field. If "Dynamic Assist Gas" is checked, the pressure will be determined by the "Near" field and "Far" field pressure 1 and pressure 2 settings on the Focus Parameters page and the current x/y position of the nozzle. When "Dynamic Assist Gas" is enabled, the Cut page does not display pressure settings because the pressure is determined by settings on the Focus page (see figure).




## DYNAMIC POWER CONTROL PARAMETERS

The **Dynamic Power Control (DPC)** function modifies average laser power as a function of cutting speed. This is accomplished by changing the resonator duty cycle. Two values on the DPC Parameters page define the power versus speed relationship: *DPC Feedrate* and *DPC Minimum Power*.

*Note: The DPC tab for the Dynamic Power Control Parameters page is visible only if "Dynamic Power Control" is selected on the Cut Parameters page.*




*DPC Feedrate:* This setting specifies the cutting speed above which the commanded duty cycle will equal the *Duty cycle* setting on the Cut Parameters page.

*DPC Minimum Power:* The duty cycle corresponding to a feedrate of zero, expressed as a percentage of the *Duty Cycle* setting on the Cut Parameters page. As the cutting speed varies from zero to the *DPC Feedrate*, the duty cycle varies proportionally from its minimum value (the cut duty cycle setting times the DPC minimum power percentage) to its maximum (the cut duty cycle setting). The average laser power at any given feedrate is the *Laser power* setting from the Cut Parameters page times the duty cycle at that feedrate.

*Example:*

| | |
|---|---|
| Laser Power: | 2500 Watts |
| Duty Cycle: | 90% |
| DPC Feedrate: | 48 IPM |
| DPC Minimum Power: | 50% |

As the feedrate approaches zero, the duty cycle approaches .9 x .5 or .45 (45%). The laser power at zero feedrate is 2500 x .45 or 1125 Watts. When the feedrate is equal to or greater than 48 IPM, the duty cycle is .9 (90%) and the power is 2500 x .9 or 2250 Watts.

This relationship is illustrated by a graph on the DPC Parameters page that shows laser power versus feedrate. Keep in mind that the "percent power" axis on this graph indicates the percentage of average power specified on the Cut Parameters page, and not the percentage of the resonator rated power. That is, 100% on this graph represents the product of the *Laser Power* and *Duty Cycle* values on the Cut Parameters page, and not full resonator rated power.

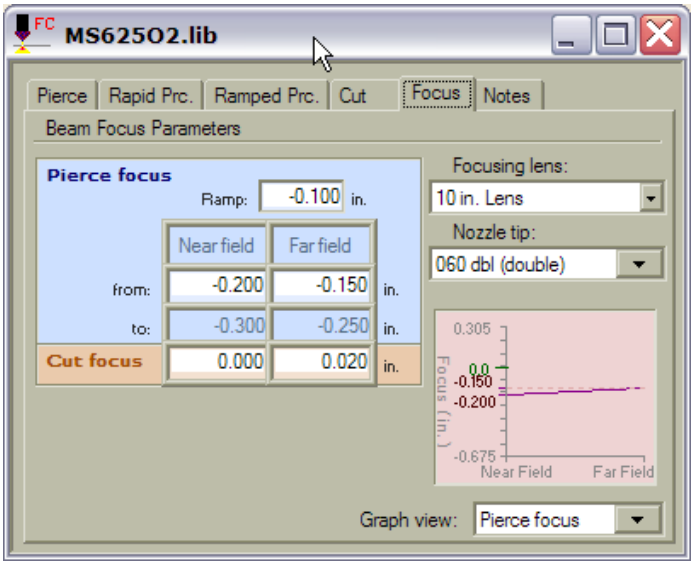
The DPC Parameters page also shows the pulse waveform for the laser beam at the minimum and maximum limits of the DPC feedrate range.

## FOCUS PARAMETERS

On the Focus Parameters page of the *Process Parameter* window, *Beam Focus position* and *Dynamic Assist Gas Pressure* parameters can be viewed and edited  This page also contains *Focusing Lens* and *Nozzle Tip* settings, which indicate which lens and nozzle are recommended for use with this material. If Machine Setup Checking is active, the lens and nozzle settings will be compared with the *Cutting Head Setup* window settings to detect machine setup errors. If Machine Setup Checking is not active, then these two fields are for reference only; the actual lens and nozzle are to be manually installed by the operator.

### Beam Focus Position

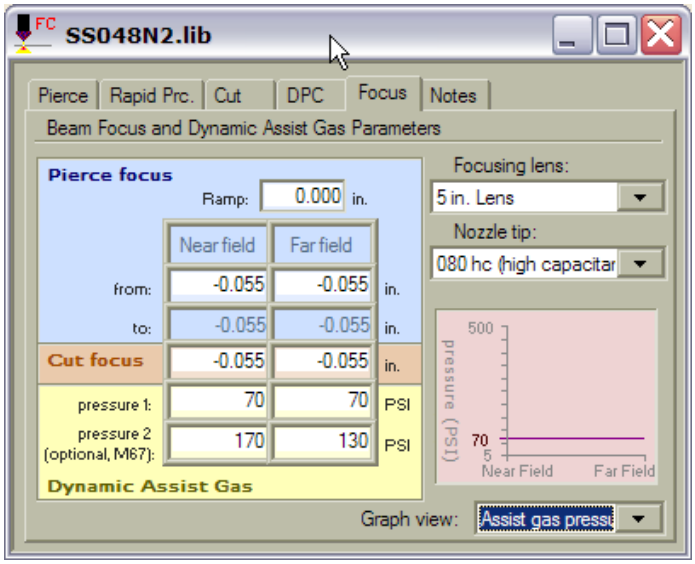
The Focus Parameters page has separate focus settings for the **Pierce and Cut** process, and each process has a *Near field* and *Far field* setting. The *Near field* setting specifies the focus position when the cutting head is closest to the laser source. The *Far field* setting specifies the focus position when the cutting head is farthest from the laser source. As the Autofocus cutting head moves along the laser beam length, it changes the focus position periodically to simulate a linear transition from the *Near field* to *Far field* setting.




The pierce focus can be configured to change (ramp) during the pierce sequence. The **Pierce** process always begins by commanding a focus position based on the *Near field* and *Far field* "from:" settings and the position of the cutting head in the X/Y plane. The *Ramp* setting is the distance that the pierce focus will increase or decrease during the pierce sequence to implement a ramped pierce focus. The process applies the same ramp change to the "from:" focus settings at the *Near field* and *Far field* positions, and to the initial focus setting calculated for any position between them. For reference, the window displays the focus positions calculated for the end of the pierce time, in the row labeled "to:".

### Dynamic Assist Gas Pressure

If the Cut Parameters page has "Dynamic Assist Gas" checked, then the Focus Parameters page has two sets of parameters for *Near field* and *Far field Dynamic Assist Gas Pressure*. As the cutting head moves along the laser beam length, the cutting assist gas pressure changes to produce a linear transition between the *pressure 1* settings at the Near field and Far field. If the program commands **M67**, the assist gas pressure changes between the *Near field* and *Far field* settings specified for pressure 2.

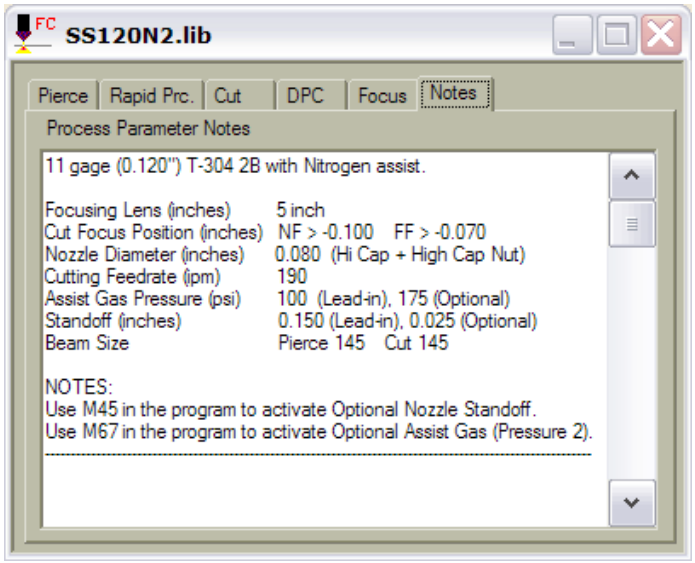



The Focus page includes a plot to show how the selected Process Parameter changes with laser beam length. When the page has more than one set of *Near field* and *Far field* parameters, the Graph view selection changes the plot to display *Pierce focus*, *Cut focus*, or *Assist gas pressure*.

*Note: The **Assist Gas Pressure Override** setting does not affect assist gas pressure when the program uses Dynamic Assist Gas.*

## PROCESS PARAMETER NOTES

The Process Parameter Notes page contains general information pertaining to the cutting process for each material type. The Process Parameter files supplied by CINCINNATI INCORPORATED typically list the recommended lens type (mounting distance), nozzle orifice diameter, and focal position on the Notes page. Other information may be added to this page, as desired, to assist the operator in cutting a specific material.


SS120N2.lib
Pierce
Rapid Prc.
Cut
DPC
Focus
Notes
Process Parameter Notes
11 gage (0.120") T-304 2B with Nitrogen assist.
Focusing Lens (inches) 5 inch
Cut Focus Position (inches) NF > -0.100 FF > -0.070
Nozzle Diameter (inches) 0.080 (Hi Cap + High Cap Nut)
Cutting Feedrate (ipm) 190
Assist Gas Pressure (psi) 100 (Lead-in), 175 (Optional)
Standoff (inches) 0.150 (Lead-in), 0.025 (Optional)
Beam Size Pierce 145 Cut 145
NOTES:
Use M45 in the program to activate Optional Nozzle Standoff.
Use M67 in the program to activate Optional Assist Gas (Pressure 2).

# PROCESS MONITOR

The Process Monitor can improve the piercing and cutting processes by monitoring the intensity of the light generated by the laser beam at the nozzle tip. Process Monitor is an optional system that requires special equipment and a *Machine Configuration Option* setting.

The following features are available only on Laser Systems equipped with Process Monitor:

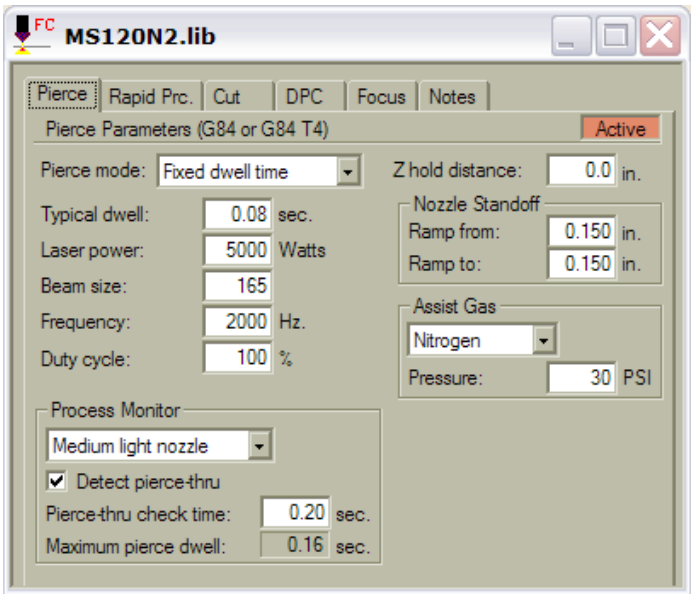
- *Pierce-Through Detection:* The pierce sequence is automatically terminated as soon as the beam penetrates the sheet.
- *Adaptive pierce mode:* Adaptive pierce avoids blowouts while maintaining a reasonably short pierce time (recommended only for oxygen piercing 1/4" and thicker).
- *Loss of Cut Detection:* Automatically stops the cutting program of the cut is lost (recommended only for nitrogen cutting 14 gage and thicker).

When the *Process Monitor* option is enabled, the Pierce Parameters page contains additional parameters grouped under the **Process Monitor** heading, and the Cut Parameters page contains additional parameters under the **Loss of Cut Detection** heading (see figures that follow).

## PIERCE-THROUGH DETECTION

When Pierce-Through Detection is used with *Fixed dwell time* pierce mode, **G84** initiates a constant power pierce sequence that will terminate when either: (1) pierce-through is detected or (2) the *Maximum pierce dwell time* period elapses.

When Pierce-Through Detection is used with *Ramped power* pierce mode, **G84** initiates a ramped-power pierce sequence based on the Ramped Pierce Parameters page settings. Pierce-Through Detection may terminate the pierce sequence before the ramp pattern is complete.



*Detect pierce-thru:* This check box enables or disables Pierce-Through Detection for Fixed dwell time and Ramped pierce power pierce modes. If this box is not checked, the pierce sequence duration is always the time specified by the *Pierce dwell* parameter. When Adaptive pierce mode is used, Pierce-Through Detection is always enabled and this check box is grayed.

*Note: When Pierce-Through Detection is active, the label for the "Pierce dwell" field changes to "Typical dwell" since the pierce time is no longer a constant value. The actual pierce dwell time may be shorter or longer than the* ***Typical dwell time****, depending on when pierce-through is detected. However, the final* ***Ramped Pierce Focus*** *and* ***Ramped Pierce Standoff*** *values will still occur at the* ***Typical dwell time****, regardless of when pierce-through is detected.*

*Nozzle type*: Nozzles are classified as either "Low light", "Medium light", or "High light". This is due to the fact that nozzle geometry affects the amount of light reaching the sensor (which is on the Z block). The nozzle type setting is used to estimate the range of expected light intensity. Nozzles are classified as follows:

| ASSIST GAS | NOZZLE | MENU SELECTION |
|---|---|---|
| Nitrogen | .120 or .160 | High light nozzle |
| Nitrogen | All other | Med. light nozzle |
| Oxygen | Double or .040 | Low light nozzle |
| Oxygen | .120 or .160 | High light nozzle |
| Oxygen | All other | Med. light nozzle |

*Pierce-through check time:* Pierce-through is detected when the measured light intensity falls below a pre-defined threshold. The light tends to flicker briefly before pierce through, particularly for thicker materials. The *check time* is an interval during which the light intensity must stay low before pierce through is assured. Typically, sparks will be visible below the sheet for this time period while the pierce hole is being cleaned out. This parameter should not be set to zero. Recommended check times are as follows:

| THICKNESS FROM | THICKNESS TO (NOT INCLUDING) | TIME (SEC) |
|---|---|---|
| Less than 0.25" | 0.25" | 0.05 |
| 0.25" | 0.5" | 0.1 |
| 0.5" | 0.75" | 0.3 |
| 0.75" | Greater than 0.75" | 0.7 |

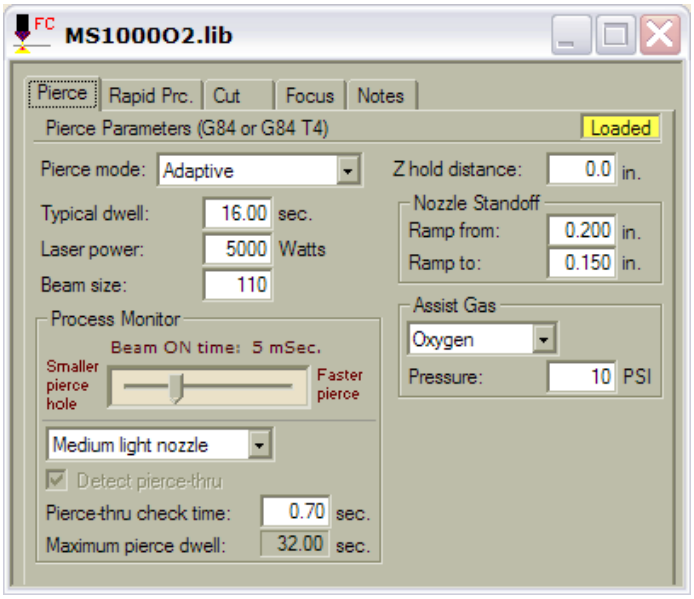
*Maximum pierce dwell:* This is the longest pierce time allowed when Pierce-through detection is used. If a pierce occurs directly over a slat, pierce-through may not be detected and the pierce sequence will terminate after this dwell time. The *Maximum pierce dwell* is always preset to a value obtained by multiplying the *Typical dwell* setting by a fixed multiplier, and therefore cannot be edited.

*Actual Pierce Time Display:* The actual pierce time is displayed in the *Process Monitor* window, which can be opened from **Maint. | Diagnostics**.

## ADAPTIVE PIERCE MODE

In Adaptive pierce mode, Process Monitor controls laser pulsing to achieve the shortest possible pierce time while avoiding blowouts. *Laser pulse ON* time is fixed at the value specified by the *Beam ON time* setting on the Pierce Parameters page. The time between pulses (off time) is controlled by the Process Monitor by delaying each new pulse until the light from the preceding pulse dies away.

A pierce blowout is indicated if the time between pulses exceeds a preset limit. When a blowout is detected, the beam and assist gas are turned off for several seconds, then the pierce is restarted. (This is similar to what happens after a tip touch, except the head stays down.) If a second blowout occurs during a pierce, the machine is stopped and an alarm message is displayed.



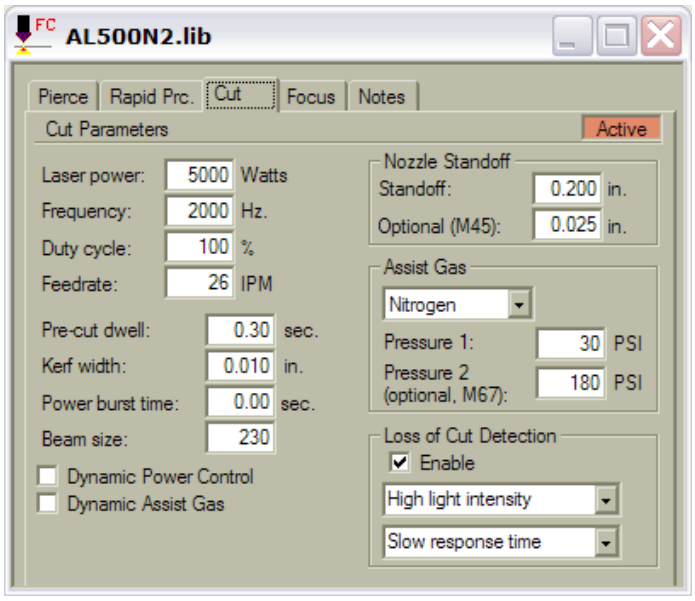

*Beam ON time:* When Adaptive pierce mode is selected, the *Frequency* and *Duty cycle* parameters are replaced with the *Beam ON time* slider control. The user can use this slider control to set the *Beam ON time* from 1 to 15 milliseconds. Adjusting the *Beam ON time* affects the pierce-through time versus pierce hole size relationship. Generally speaking, increasing the *Beam ON time* makes the pierce more aggressive. (i.e., shorter time, larger hole, larger pierce mound), and increases the possibility of a blowout.

*Pierce-thru check time:* See description in the **PIERCE-THROUGH DETECTION** topic.

*Maximum dwell:* See description in the **PIERCE-THROUGH DETECTION** topic.

## LOSS OF CUT DETECTION

Uncontrolled loss of cut during nitrogen cutting causes a flash of plasma. When the Process Monitor detects this condition, the **Loss of Cut Detection** function will quickly stop the cutting process to minimize material damage. **Loss of Cut Detection** is only recommended for use when cutting with nitrogen assist gas.




*Enable:* Use this check box to enable or disable the **Loss of Cut Detection** function.

*Light intensity:* This pull-down list determines the light threshold corresponding to loss of cut. High light intensity is used for materials that generate plasma during normal cutting. This would include 16 gage and thinner mild steel, and .125 inch and thinner stainless. Low light intensity is used for materials that generate little or no plasma during normal cutting and use .040 inch or double nozzles. Generally speaking, lower thresholds increase the odds that a lost cut will be detected, and also make loss of cut indications trips more likely. Higher thresholds reduce spurious indications but may result in loss of cut not being detected.

*Response time:* Some materials, notably aluminum and stainless, are prone to brief flashes of plasma during normal cutting. Use the slow response time (roughly .4 second) for these materials. For other materials use the fast response time (roughly .1 second). The slower response time will cut down on spurious loss of cut detections.

*Z Hold Distance:* Note that there is often a flash of plasma at the start of the cut, immediately after the pierce. To prevent spurious loss of cut detections during this time, set the *Z hold distance* parameter on the Pierce Parameters page to a non-zero value. Loss of cut detection is disabled while the nozzle tip is within the Z hold distance zone.

# START-UP AND SHUT-DOWN

## LASER SYSTEM START-UP

1. Perform the steps in the **SAFETY MAINTENANCE CHECK** portion of **SECTION 3 – SAFETY,** and any steps that do not require operating the machine in the **DAILY MACHINE INSPECTION** portion of **SECTION 9 – MAINTENANCE AND ADJUSTMENTS** of the Laser System **Operation, Safety, and Maintenance Manual**.

2. Turn on the Assist Gas supplies. Verify that they are regulated to the required pressures.

   *Note: The supply pressure must be about 100 PSI (690 kPa) above the maximum desired cut pressure.*

3. Turn on the Laser Gas Supply. Pressure to the laser center should be about 28 PSI.

4. Verify that the main electrical disconnect at the rear of the main frame is on. If the electrical supply to the chiller has been off for 30 minutes or less, then verify that the chiller power disconnect is on and proceed to the next step. If the chiller disconnect has been off for more than 30 minutes, then follow the instructions in this CAUTION note:

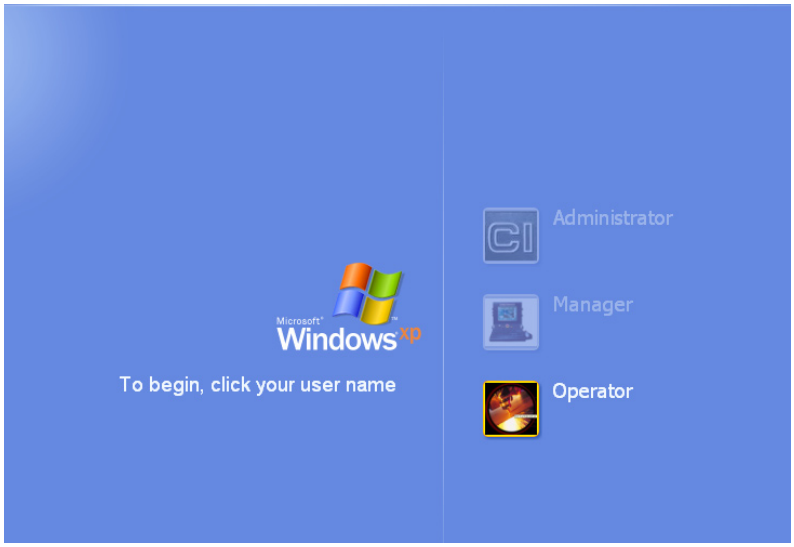
**CAUTION**

**If chiller power disconnect has been off for more than 30 minutes, turn the chiller selector switch to STANDBY, turn the chiller disconnect ON and then wait 8 hours before turning the selector switch to LOCAL or REMOTE. Failure to follow this procedure can damage the chiller. CINCINNATI INCORPORATED recommends leaving the chiller power disconnect ON at all times, unless maintenance or repair procedures require turning off chiller power. Refer to the "CHILLER WARM-UP" section for more details.**

5. Verify the chiller temperature control set-point is 80.0°F (26.7°C). To start and stop the chiller with the "Laser Mains" button on the touchscreen, turn the chiller selector switch to "REMOTE". To run the chiller independent of the Laser System control, turn the chiller selector switch to "LOCAL". In Local mode, the chiller should start immediately.

6. To operate the Laser System, log on to the Microsoft Windows® - based PC control to start the Laser System **CNC/HMI Software** application. If the previous user did not log off, then the **CNC/HMI** program should be running, proceed to Step 7. If the screen saver is displayed on the control station monitor, then move the trackball or press any keyboard key to restore the *CNC/HMI Software Interface* display. If the CNC/HMI user interface is not visible, then proceed with the following instructions to log on.

   When the control is first powered-up, the Windows Logon screen will be displayed on the control station monitor. To log on, click a user name and enter the corresponding password. There are different user logon levels, starting with the Operator user.




## User Logon Levels

*Operator:* For normal machine operation, select "Operator" for the user name and enter "driver" for the password (without quotation marks). The Laser System **CNC/HMI** application will start automatically after each Operator logon. The Operator user has access to all user interface controls needed to prepare the Laser System for operation, load and execute cutting programs. Operator has limited access to *Machine Configuration* settings.

*Note: The user name is NOT letter case sensitive. Any combination of upper or lowercase letters can be entered for the user name. The password **is** case sensitive. The user must enter the appropriate password exactly as instructed.*

*Manager:* The Manager level extends Operator-level access by adding the capability to perform administrative functions such as installing software updates, setting the system date/time, backing-up/restoring files and configuring the automatic backup system. See the Windows Administration topic for more about these administrative functions. The Manager logon level also provides edit access to all *Machine Configuration* parameters.

*Setup:* The Setup level combines the administrative capabilities of the Manager level with network/domain setup and configuration capabilities. The Setup level is limited to performing administrative functions. The laser cutting system can not operate using the Setup logon level. To log on to the Setup account, press the "**Ctrl + Alt + Delete**" key combination twice while the Windows Logon screen is displayed. This will cause a login box to open, enter "setup" for the user name and the corresponding password.

1. If the **CNC/HMI** program is not already running, start it now by clicking the 'CNC LASER' icon.

2. Prepare the resonator for operation by pressing the "Laser Mains" button on the Operator Console Control Bar. If the chiller is in REMOTE mode, verify the chiller starts. The resonator control will start a gas purge cycle that takes about 3 minutes to complete. Open the *Laser Status Monitor* window. During the purge cycle, the "Laser Mains" button will be flashing and "Operating State" field in the *Laser Status Monitor* window will display "Wait…". While the purge cycle is in progress, continue with the next two steps.

3. When the **CNC/HMI** application begins running, the *Alarms and Messages* window will display: "Main drives are off", "Axes have not been homed", "Tip standoff calibration not completed", and "Auto Focus Head not homed". Make sure all personnel and equipment remain clear of the moving gantry. Turn the "DRIVES" selector keyswitch on the Operator Control Station Side Panel to the "ENABLED" position, press the "DRIVES ON" pushbutton and then press the "RESET" button. The "Drives On" indicator on the Control Station Side Panel should illuminate and the *Main drives are off* message should disappear from the *Alarms and Messages* window.

4. Home the machine axes by selecting the "Axes Home" button on the Operator Console Control Bar and then pressing the "CYCLE START" pushbutton. When the homing operation starts, the "Axes Home" button text changes to "Homing" and the color changes to amber. When all axes have moved to their reference positions, all axis motion stops, the **Axes Home** function ends, and the control automatically changes to Jog mode.

5. Wait until the Mains startup sequence resonator purge cycle is complete. This is indicated when the "Operating State" field in the *Laser Status Monitor window* displays "Ready" and the "Laser Mains" button is no longer flashing. With the resonator in the Ready state, rotate the "HIGH VOLTAGE" keyswitch on the Control Station Side Panel to the "ENABLED" position, and then press the "High Voltage" button on the Operator Console Control Bar. The resonator control will now start a power calibration sequence that may take several minutes to complete. While the resonator is calibrating, the "High Voltage" button will be flashing and "Operating State" field in the Laser Status monitor window will display "Calibrating". When the resonator power calibration sequence is complete, the "High Voltage" button will be bright green and not flashing, and the "Operating State" field in the *Laser Status Monitor* window will displays "Run". While the calibration sequence is in progress, perform the safety checks in the next two steps.

6. Perform the steps in the **SAFETY MAINTENANCE CHECK** portion of **SECTION 3 – SAFETY,** and in the **DAILY MACHINE INSPECTION** portion of **SECTION 9 – MAINTENANCE AND ADJUSTMENTS** of the **Laser System Operation, Safety, and Maintenance Manual** that require operating the machine control.

7. To begin cutting, see the **CUTTING PROCEDURES** topic.

## CHILLER WARM-UP

The chiller compressor contains a heater to maintain compressor oil temperature and prevent refrigerant condensation while the chiller is not running. Since the chiller is typically powered from the machine main power disconnect, the machine main power disconnect and the chiller disconnect should normally be left on.

If the machine main power disconnect or the chiller disconnect is off for more than 30 minutes, refrigerant condensation may occur in the compressor. If this condensation occurs, immediate startup of the chiller would reduce compressor life. To ensure maximum compressor life, the compressor oil should be preheated for 8 hours after a power loss of 30 minutes or more. To preheat the chiller, verify the chiller "SYSTEM" switch is at "STANDBY", turn the main disconnect and chiller disconnect on, and wait 8 hours before running the chiller. The "SYSTEM" switch may then be turned to "LOCAL" or "REMOTE" for normal operation. Note that the heater in the chiller compressor operates whenever power is supplied to the chiller.

## SHUTTING DOWN THE LASER SYSTEM

1. Turn the "SHUTTER" keyswitch on the Operator Control Station Side Panel to the "LOCK/CLOSED" position.

2. Switch off the laser high voltage power supply by selecting the "HIGH VOLTAGE" button on the Operator Console Control Bar, and then rotate the "HIGH VOLTAGE" keyswitch on the Control Station Side Panel to the "LOCK/OFF" position. The "Laser Mains" button on the Operator Console Control Bar will now begin flashing. Wait for the "Laser Mains" button to stop flashing; This should take about 20 seconds.

3. Switch off the laser MAINS power supply by selecting the "Laser Mains" button on the Operator Console Control Bar. This action starts the laser gas purge cycle. Open the *Laser Status Monitor* window. During the purge cycle, the "Operating State" field in the *Laser Status Monitor* window will display "Wait". The laser gas purge takes about 15 minutes. While waiting for laser purge to finish, continue with the next two steps.

4. Turn off the assist gas supplies and press each assist gas button to bleed off the assist gas pressure (one at a time). Assist gas buttons are displayed in the Auxiliary Functions Control Bar.

5. Switch off the main drives by rotating "DRIVES" keyswitch on the Control Station Side Panel to the "LOCK/OFF" position.

6. Wait until the laser gas purge cycle is complete. This is indicated when the "Operating State" field in the *Laser Status Monitor* window changes from "Wait" to "Purged" and the light in the "Laser Mains" button is no longer bright green. Now turn off the laser gas supply.

7. With the laser mains off and the main drives off, the chiller will shut down automatically if the chiller switch is in the "REMOTE" position. If the chiller is in Local mode, switch it to "STANDBY".

8. Terminate the **CNC/HMI** application using any of the application **Exit** commands.

   *Note: CINCINNATI INCORPORATED recommends leaving the machine disconnect on at all times unless maintenance or repair procedures require turning off power to the machine. If no such maintenance or repairs are required, the above steps complete a normal shutdown. If machine disconnect must be turned off, proceed to the next step.*

9. If the main power disconnect is turned off, be sure to wait for the Operator Control Station screen to go completely blank before servicing the Laser System computer. Since the computer is powered by an uninterruptible power supply (UPS), power to the computer will remain on for several minutes after the disconnect is turned off. After the main power disconnect is turned off, the operating system will begin the automatic shutdown procedure. When automatic shutdown is complete, the Operator Control Station screen will be blank, indicating the UPS has turned off power to the computer.

**CAUTION**

**If chiller power disconnect has been off for more than 30 minutes, turn the chiller selector switch to STANDBY, turn the chiller disconnect ON and then wait 8 hours before turning the selector switch to LOCAL or REMOTE. Failure to follow this procedure can damage the chiller. CINCINNATI INCORPORATED recommends leaving the chiller power disconnect ON at all times, unless maintenance or repair procedures require turning off chiller power. Refer to the "CHILLER WARM-UP" section for more details.**

10. To prevent unauthorized machine operation, remove keys from all key-operated control switches.

    *Note: Some keys can operate more than one switch.*

**! ! DANGER ! !**

**THE LASER SYSTEM COMPUTER HAS A "UPS" (UNINTERRUPTABLE POWER SUPPLY) FOR POWER PROTECTION. AFTER MAIN POWER SHUTDOWN, THE POWER TO THE COMPUTER WILL REMAIN ON FOR UP TO 10 MINUTES BEFORE SHUTTING DOWN.**

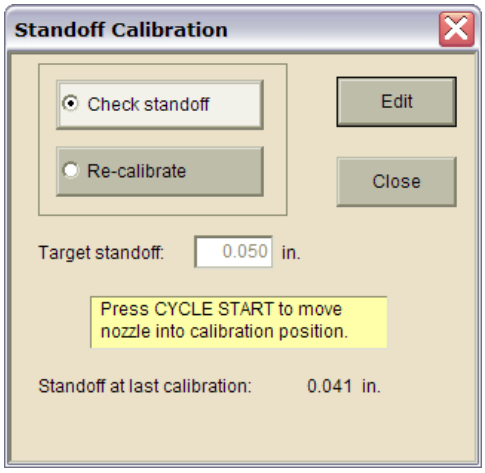
# CALIBRATION AND ADJUSTMENT PROCEDURES

## NOZZLE STANDOFF CALIBRATION

*Note: If the standoff sensor loses accuracy quickly and requires frequent calibration, see the **Noncontact Standoff Sensing Accuracy** topic.*

*Note: This procedure applies to the noncontact cutting head without the optional Tactile Foot. For the Tactile Foot option, use the **Standoff Calibration** procedure in the **Tactile Foot** Manual Supplement (EM-483).*

1. To calibrate the nozzle standoff select the "Standoff Calibration" tool bar button or select the **Standoff Calibration** command from the *Utilities* menu to open the *Standoff Calibration* window. The laser control is now in the "Special Operating Mode" used for noncontact head standoff calibration.




2. Select "Edit" and then enter the desired cutting standoff in the "Target Standoff" field.

3. Jog the cutting head over a firmly secured piece of clean conductive material, at least 8.0 by 8.0 inches (200 mm by 200 mm). Material used for Standoff Calibration can have a different thickness than material used for laser cutting.

   *Note: For Standoff Calibration to work properly, the surface must be conductive. If the (conductive) material has a protective plastic coating, peel back the coating temporarily until calibration is completed.*

4. To check the standoff accuracy without making any changes, select the "Check Standoff" button. To recalibrate from scratch, thereby zeroing all internal drift compensation values, select the "Recalibrate" button.

5. Press the "CYCLE START" button to initiate the calibration process. The cutting head will move down until the nozzle tip touches the workpiece and then it will retract to the desired standoff. When the move has completed, the status window will read: "Nozzle is in calibration position".

### NONCONTACT STANDOFF SENSING

Any CINCINNATI Laser System equipped with a noncontact head includes a capacitive sensor as part of the Z-axis positioning system. This sensor produces an electrical signal that accurately corresponds to standoff by measuring very small changes in the electrical capacitance between the nozzle tip assembly and the workpiece. The tip performs two functions: it directs the assist gas and serves as half of a capacitor whose capacitance changes with standoff. In order to perform these functions, nozzle shape and cleanliness are critical.

Contamination or damage to the nozzle tip assembly can cause the following problems:

- Sudden or gradual drift in standoff calibration.
- Z-axis instability.
- Sudden Z-axis movements or following error alarms.
- Spurious tip touch or sensor failure alarms.
- Frequent need to change cutting parameters.

### Nozzle Tip Assembly

The nozzle tip assembly consists of the nozzle tip, tip retainer nut, the tip retainer, outer cone and several internal insulators. (See Cutting Head Assembly Diagram ). The tip retainer, tip retainer nut and nozzle tip, which are all copper, form one "plate" of the capacitor whose capacitance is measured by the sensor. The tip retainer and outer cone are electrically insulated. If contamination forms a conductive path between these two components, drift or spurious tip touch alarms will result.

Keep the nozzle tip assembly clean. Cutting debris, condensed vapor, oil, or lens coolant can compromise sensor accuracy. It is good practice to clean this assembly weekly. Cutting materials that are oily or produce vapors may necessitate more frequent cleaning. Cleaning procedures and materials are critical. See **NONCONTACT HEAD CLEANING** in **SECTION 9 – MAINTENANCE AND ADJUSTMENTS** of the **Laser System Operation, Safety, and Maintenance Manual** for more on nozzle tip assembly maintenance.

## LASER SHOT

The lens centering procedure described later in this section, the beam delivery alignment procedure described in **SECTION 9 – MAINTENANCE AND ADJUSTMENTS** of the **Laser System Operation, Safety, and Maintenance Manual**, and other maintenance tasks require taking a "laser shot" numerous times. A laser shot opens the shutter and turns the beam on for a specified period of time at a specified power level. The laser shot also opens the nitrogen assist gas valve to protect the lens.

The laser shot has several other names depending on the application. When the laser shot is used for lens centering, it is called a "tape shot" because the beam passes through a section of masking tape attached to the cutting head nozzle. When the laser shot is used for beam delivery alignment, it is called a "card shot" because the beam strikes a target card. When the laser shot is used to determine the characteristics, or mode, of the laser beam, it is called a "mode shot" or "mode burn". The laser shot is often referred to as a "shutter flash" since the shutter is opened briefly.

*Note: The procedure for taking a mode shot is not described in the* **Laser System Operation, Safety, and Maintenance Manual***. The mode shot procedure is described in laser maintenance training.*

### SETUP FOR A LASER SHOT

To take a laser shot, open the *Mirror Alignment* or *Lens Centering* window. For card shots and mode shots, use *Mirror Alignment* window. For tape shots, use the *Lens Centering* window.

When the *Mirror Alignment* or *Lens Centering* window opens, the user can verify and/or edit the settings. This table shows typical settings for each type of alignment shot:

| | LENS CENTERING (TAPE SHOT) | BEAM ALIGNMENT (CARD SHOT) | MODE EVALUATION (MODE SHOT) |
|---|---|---|---|
| Power | 50 W. | Max. rated power * | Max. rated power * |
| Frequency | N/A | 50 Hz. * | 2000 Hz * |
| Duty Cycle | N/A | 3 % * | 100 % * |
| Length of Flash | 0.100 sec. | 0.020 sec. * | 2.000 sec. * |
| Beam Size | Cut beam size from Process Parameter file for current material | 170 * | 185 * |
| * Typical values are shown here. See **SECTION 9 – MAINTENANCE AND ADJUSTMENTS** of the **Laser System Operation, Safety, and Maintenance Manual** for settings corresponding to specific resonators. | | | |

## TAKING A LASER SHOT

With the *Mirror Alignment* or *Lens Centering* window open, enable the Remote Station by rotating the keyswitch on the Remote Station to the "STATION ENABLE" position. Also, make sure the "SHUTTER" keyswitch on the Control Station Side Panel is in the "ENABLE" position.

**! ! DANGER ! !**

**KEEP HANDS AWAY FROM BEAM PATH DURING A LASER SHOT.**

To take a laser shot, rotate and hold the "ARM SHUTTER" switch 1/8 turn clockwise and then press the "SHUTTER FLASH" pushbutton on the Remote Station.

When the "SHUTTER FLASH" pushbutton is pressed, the shutter will open and the laser beam will turn on at the specified power level and duration. As soon as the "SHUTTER FLASH" pushbutton is released, the shutter flash will end, even if the beam has not been on for the full time period set in the *Mirror Alignment* or *Lens Centering* window.

# CENTERING THE FOCUSING LENS

### Background

In order to obtain clean cuts with minimum dross on the bottom side of the cut edge, the focused beam must be centered in the nozzle tip orifice. Slight misalignment will cause dross (slag) to adhere to the bottom side of the cut, on either the part or the parent sheet. Severe misalignment can cause the beam to "clip" the copper nozzle tip and damage it. The lens or cutting head can also be damaged if this condition persists.

### Prerequisites

- Machine startup procedure is complete
- High voltage is on
- Nozzle tip is installed and in good condition

### Materials Required

- Masking tape
- Ink pad or felt tip marker (Mark-X brand works well)
- Magnifying lens (7X recommended)
- Alignment tip (C.I. #921915)

## LENS CENTERING PROCEDURE

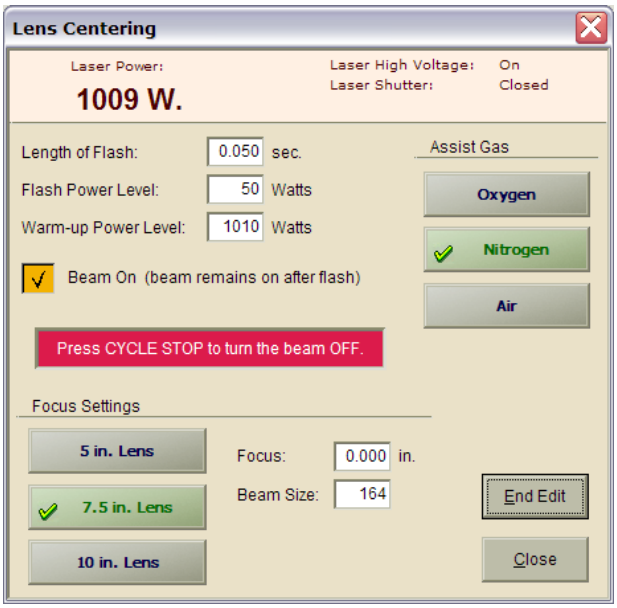
1. If the lens is not already in its spring-loaded holder, insert the lens in the holder (curved side of lens up, flat side down) and tighten securely with the provided lens insertion tool. The curved side of the lens can be identified by noting that reflected images from the curved side (convex) appear smaller than reflected images from the flat side. Inspect both sides of the lens for dirt, and clean the lens if necessary. See **OPTICS HANDLING AND CLEANING** in **SECTION 9 – MAINTENANCE AND ADJUSTMENTS** of the **Laser System Operation, Safety, and Maintenance Manual** for lens cleaning procedures.

   *Note: A lens can be easily damaged if mishandled. Refer **OPTICS HANDLING AND CLEANING** in **SECTION 9 – MAINTENANCE AND ADJUSTMENTS** of the **Laser System Operation, Safety, and Maintenance Manual** for optics handling procedures.*

2. Insert the lens holder in the cutting head. Make sure the lens holder is inserted with the seal down (curved side of lens up). Push the lens holder into its slot against the spring pressure and tighten the locking screw to lock the holder in place.

3. Momentarily activate nitrogen assist gas to force the cutting head components into proper alignment. To activate nitrogen assist gas, open the Auxiliary Functions Control Bar and select the "Nitrogen" button. After one second, select the button again to stop gas flow.

4. Open the *Lens Centering* window. Recommended values are Length of Flash = 0.050 seconds and Flash power level = 50 Watts. Click "Edit" to change any settings in the *Lens Centering* window. Note that changed settings will not take effect until "End Edit" is selected.




**CAUTION**

**Do not use oxygen assist gas for the lens centering procedure. The lens can be permanently damaged. Make sure that an inert gas such as nitrogen is used for this procedure.**

*Note: The resonator can be warmed-up before the tape shot, for a better simulation of operating conditions. See the **Lens Centering Window** topic for more on enabling resonator warm-up for the **Lens Centering** function.*

5. Apply a small amount of ink to the nozzle tip by pressing an ink pad against the tip or by rubbing the tip with a felt marker pen. Place a length of masking tape in a known orientation over the orifice in the nozzle tip. Press the tape firmly on the tip.

6. Unlock the shutter by turning the "SHUTTER" keyswitch on the Control Station Side Panel to "ENABLED".

**! ! DANGER ! !**

**KEEP HANDS AWAY FROM BEAM PATH DURING A LASER SHOT.**

7. Take a laser shot. (See the **Laser Shot** topic)

8. Lock the shutter by rotating the "SHUTTER" keyswitch to the "LOCK/CLOSE" position.

9. Remove the masking tape, observing the orientation of the tape.

10. Verify centering of the beam in the orifice. View the burn and the impression of the orifice on the tape with a magnifying glass. A very small burned hole should appear within the impression of the orifice (a burn hole inside a white circular area inside an ink colored donut). The objective is to center the burn hole as perfectly as possible in the white area. If the burn hole is round and requires centering, proceed to Step 11.

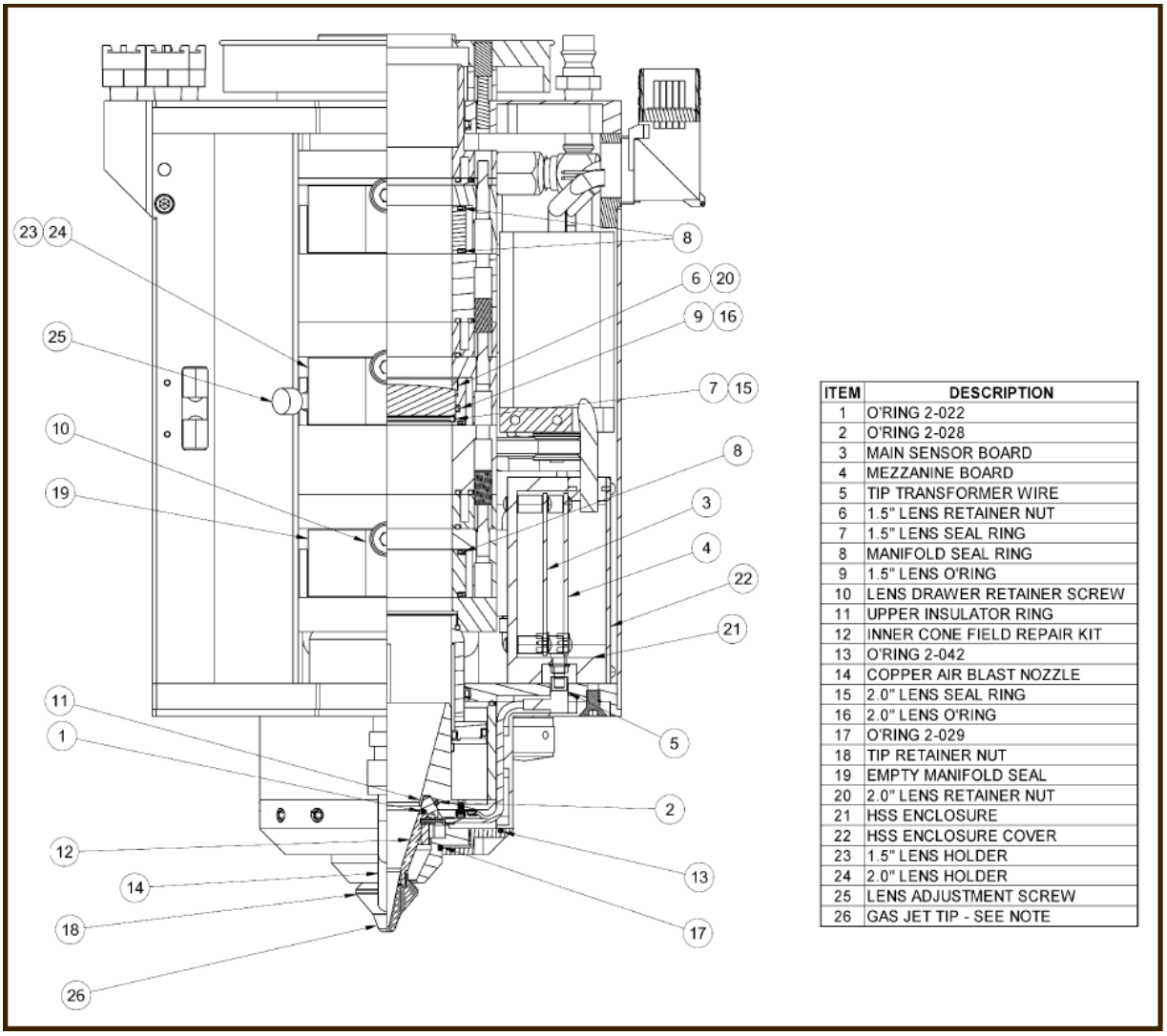
    If the burn hole is large, check the following possible causes:

    - Focus is adjusted to an extreme position.
    - Power level or length of flash is too large.
    - Lens is damaged or upside down.
    - The lens selected in the *Lens Centering* window does not match the lens installed in the field.

    If the burn hole appears as an out-of-round hole, a burned slit in the tape, or no burn hole is present, the beam may be clipping the inner nozzle wall. In this case, use the alignment tip (with a larger orifice) for the initial adjustment and repeat Steps 5 thru 10.

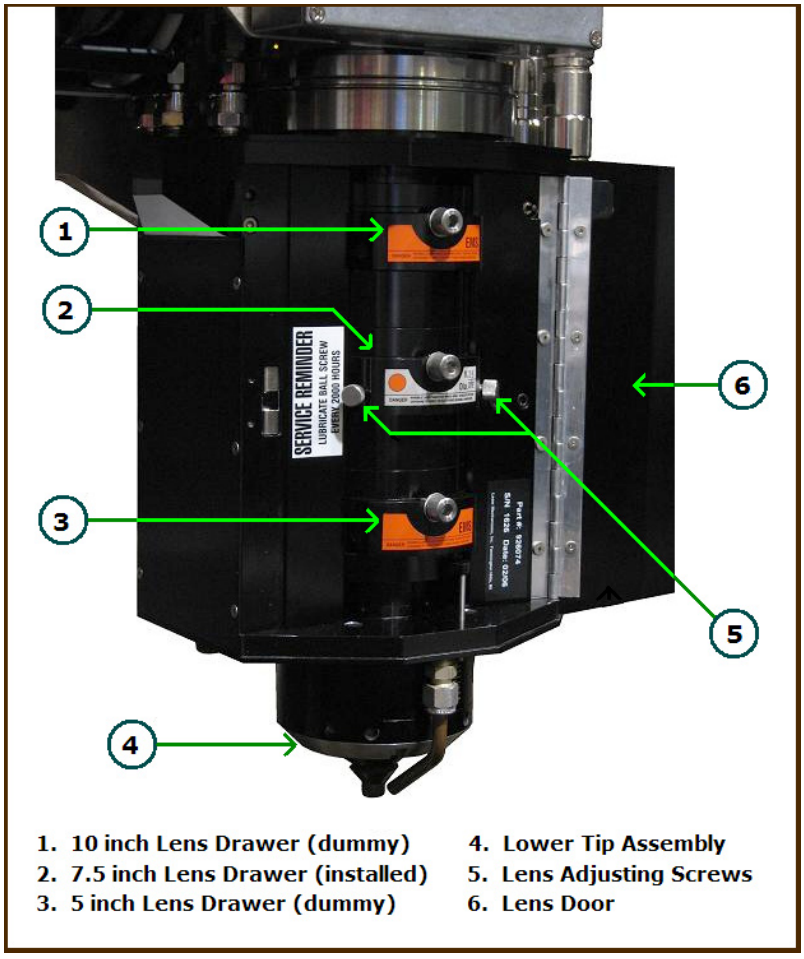
11. Adjusting the lens horizontal position will move the burn in the same direction. Find and turn the lens adjusting screws (see the *Cutting Head Assembly Diagram* and *Auto Focus Cutting Head* photograph that follows). Turning either screw clockwise will move the lens away from the screw. Turning either screw counterclockwise will move the lens toward the screw. When adjusting a screw counterclockwise, turn the screw an extra amount (CCW) and then finish clockwise (CW). This helps assure the lens will not move later. The screws have 80 threads per inch, so one turn moves the lens approximately .013 inch (0.33 mm). Since the screws are 90° apart, it may be necessary to adjust both screws to move the lens in the required direction.

12. If the alignment tip was used for initial alignment, remove it and install the appropriate nozzle tip for cutting. Repeat Steps 5 thru 11 with the cutting tip.




| ITEM | DESCRIPTION |
|---|---|
| 1 | O'RING 2-022 |
| 2 | O'RING 2-028 |
| 3 | MAIN SENSOR BOARD |
| 4 | MEZZANINE BOARD |
| 5 | TIP TRANSFORMER WIRE |
| 6 | 1.5" LENS RETAINER NUT |
| 7 | 1.5" LENS SEAL RING |
| 8 | MANIFOLD SEAL RING |
| 9 | 1.5" LENS O'RING |
| 10 | LENS DRAWER RETAINER SCREW |
| 11 | UPPER INSULATOR RING |
| 12 | INNER CONE FIELD REPAIR KIT |
| 13 | O'RING 2-042 |
| 14 | COPPER AIR BLAST NOZZLE |
| 15 | 2.0" LENS SEAL RING |
| 16 | 2.0" LENS O'RING |
| 17 | O'RING 2-029 |
| 18 | TIP RETAINER NUT |
| 19 | EMPTY MANIFOLD SEAL |
| 20 | 2.0" LENS RETAINER NUT |
| 21 | HSS ENCLOSURE |
| 22 | HSS ENCLOSURE COVER |
| 23 | 1.5" LENS HOLDER |
| 24 | 2.0" LENS HOLDER |
| 25 | LENS ADJUSTMENT SCREW |
| 26 | GAS JET TIP - SEE NOTE |

***Cutting Head Assembly Diagram***




*Auto Focus Cutting Head*

# LENS FOCAL POINT LOCATION

## Background

The diameter of the laser beam remains essentially constant as it travels through the beam delivery system. Beyond the focusing lens in the cutting head assembly, however, the beam diameter decreases with distance to a minimum at the focal point. Since the total energy of the incoming "raw" beam is concentrated in the small area of the focused spot, the resulting power density is high enough to vaporize or melt most materials. Since power density is power level divided by the area of the spot, the maximum power density occurs at the focal point (where the beam diameter is smallest). The beam diameter increases beyond the focal point of the lens, unless the beam is absorbed by some material.

The vertical location of the laser beam focal point is a critical parameter that affects the quality and accuracy of the cutting process. The focal point is different for every lens due to manufacturing and mounting variations.

## Prerequisites

- Nozzle standoff calibration is complete.
- The focusing lens has been centered.

## Materials Required

- One sheet of 16 gage (.060 inch or 1.5 mm) mild steel, at least 12 inch by 12 inch (305 mm by 305 mm), clamped in place on either pallet.
- One set of feeler gages with .004 inch to .020 inch gages in .0005 inch increments (0.10 to 0.50 mm in 0.01 mm

increments).

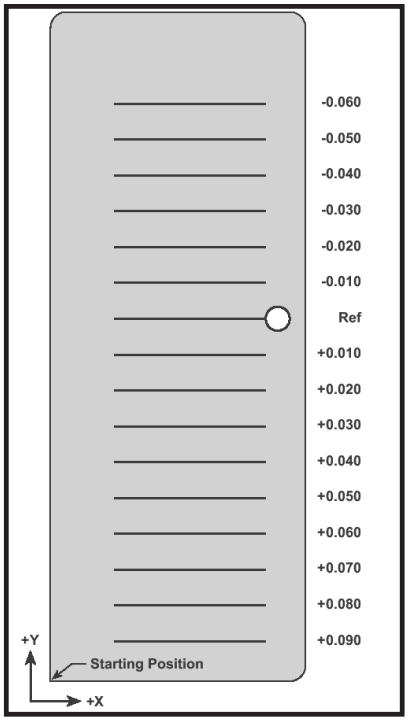
## FOCUS CALIBRATION - AUTO FOCUS HEAD

### Finding the Focus Home Setting

This procedure determines the *Auto Focus Home Offset* value that will obtain the minimum kerf in the Far field at 0.000 inch focus setting. Program "*FocusTestAuto.cnc*" cuts a series of kerf cuts in a test part, with each cut preceded by a **G102** call commanding a different focus setting. Each kerf cut corresponds to a 0.010 inch change in focal position.

### Procedure

1. Load a piece of 16 gage mild steel in the Far field of the pallet. The Far field is where the cutting head is farthest from the resonator.
2. Load the program "*FocusTestAuto.cnc*" (from the "*Program/Utilities"* directory).
3. Jog the cutting head to the lower left corner of a 3.5 x 9 inch (90 x 230 mm) area for the test part (see *Auto Focus Calibration Test Part* figure that follows).
4. Run the program.
5. The program cuts a hole at the end of one kerf cut. That kerf cut uses the zero focus position set by the *Auto Focus Home Offset* value in the *Z-axis Setup* window.
6. Use a feeler gage to measure the width of each kerf cut and then find the group of cuts with the smallest kerf widths. If the kerf cut marked with a hole is at the center of that group, then the *Home Offset* value is correct and this procedure is complete. If another cut is at the center of the group, mark it "Min Kerf" and continue with the next step.
7. Compare the test part with the focus calibration test part figure. Find the kerf cut in the figure that corresponds to the minimum kerf position on the test part. Change the *Home Offset* value by the amount shown in the figure for that kerf cut. Make sure to change the *Auto Focus Home Offset* for the currently installed lens.
8. After changing the auto focus home offset value, press "OK" to close the *Operator Set-up* window. Cut another test part to verify the "Min Kerf" cut is now the cut marked with a hole.




*Auto Focus Calibration Test Part (with Home Offset change in inches)*

## ASSIST GAS PRESSURE ADJUSTMENT

The Laser System has programmable assist gas pressure. Assist gas pressure at the cutting head tip can be programmed up to 500 PSI (3447 kPa) to suit different processing requirements. Assist gas is supplied to the machine at high pressure and flows through a pressure control valve. Pressure is adjustable up to the supply limit (minus the system pressure drop) or 500 PSI, whichever is lower.

To manually turn on the assist gas, open the Auxiliary Functions Control Bar and press the desired assist gas button/indicator. Press the button again to stop the gas flow.

**CAUTION**

**Verify correct installation of lens drawer and nozzle tip before activating either assist gas solenoid valve. Applying assist gas pressure without proper installation of these parts can force the lens drawer or nozzle tip out of position or out of the cutting head. Unrestricted flow can also damage assist gas supply components.**

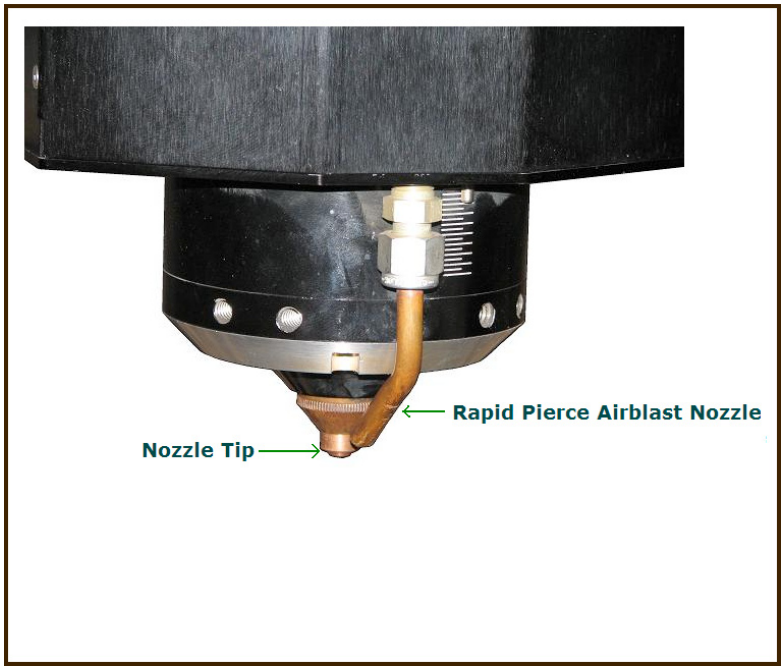
***IMPORTANT: Regulated gas supply pressure must not exceed 550 PSI (3792 kPa). Overpressure will cause damage to downstream components.***

If the assist gas pressure drops below the desired pressure setting, the control displays a system alarm message. To protect the lens, this alarm prevents continued operation of the machine.

## RAPID PIERCE ADJUSTMENTS

The optional *Rapid Pierce* feature incorporates an airblast system that operates from a supply of dry shop air. To use Rapid Pierce for oiled material, a shop air pressure of 60 to 90 PSI is adequate (400 to 600 kPa). Using Rapid Pierce with non-oiled material is not recommended. Airblast supply pressure must not exceed 250 PSI (1724 kPa).

Other than the air supply pressure, the only airblast adjustment is the position of the airblast nozzle (see picture). The mounting bracket for the airblast nozzle is designed to aim the airblast just below the bottom of the cutting head nozzle tip. The distance from the nozzle tip to the end of the airblast nozzle should be about 1 inch (25 mm).




**! WARNING !**

**A FIRE HAZARD EXISTS WHEN HIGH TEMPERATURE MATERIAL IS EJECTED FROM THE PIERCE ZONE. THE MACHINE MUST NOT BE LEFT UNATTENDED WHILE USING THE RAPID PIERCE OPTION.**

# CUTTING PROCEDURES

## FIRST RUN OF PARTS

Before cutting a part or nest of parts for the first time on the laser, complete these steps:

1. **Load Workpiece**

   Place the control in Jog mode by pressing the "Jog" button on the Operator Console Control Bar. Load the workpiece on the pallet against the X- and Y-stops and clamp the workpiece in place. Make sure the surface of the workpiece is clean and free of rust and scale. Remove any tape or paint in the cutting area of the workpiece (top and bottom).

2. **Move Pallet In**

   Use the "PALLET" pushbuttons on the Operator Control Station front panel to move the pallet to the cutting position. If a second pallet is in the cutting position, move it to the OUT position.

3. **Load Program**

   Place the control in Auto mode by pressing the "Auto" button on the Operator Console Control Bar. Load a cutting program. (See **FILE MENU** commands **LOAD NC PROGRAM** and **LOAD CURRENT PROGRAM**.)

4. **Check Process Parameters**

   Open the Process Parameter file and verify that the cutting parameters are correct for the material to be cut. (See **FILE MENU** commands **OPEN** and **OPEN CURRENT PROCESS.**)

5. **Check Machine Adjustments**

   Depending on the circumstances, some or all of the adjustments listed next should be performed. When setting up for the first cutting job of the day, perform all of the adjustments. If setting up to cut a different material, check any adjustment that needs to change. If setting up to cut more of the same material but with a different program, no adjustments are required. If subsequent cutting performance is not satisfactory, check all of these adjustments.

   *Nozzle tip:* Remove the nozzle tip and verify that it is clean and undamaged. Clean or replace as necessary. Use the nozzle tip specified on the Focus page of the Process Parameter file, or in the **Process Parameters and Diagnostic Guide**.

   *Standoff:* Check the standoff calibration (see **Nozzle Standoff Calibration**).

   *Focusing lens:* Remove and inspect the focusing lens. Clean or replace the lens as necessary. Use the lens specified on the Focus page of the Process Parameter file, or in the **Process Parameters and Diagnostic Guide**.

**CAUTION**

**If the lens holder is not securely installed, assist gas pressure can force the lens holder out of the cutting head. Push the lens holder into the cutting head and tighten the locking screw. Secure an empty manifold seal (EMS) in the unused lens locations.**

   *Lens centering:* Center the lens (see **Centering the Focusing Lens**).

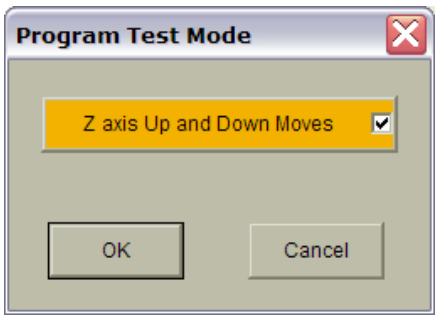
## 6. Run Program

To run the loaded program, turn the "SHUTTER" keyswitch on the Operator Control Station Side Panel to the "ENABLED" position, verify high voltage is on, and then press the "CYCLE START" pushbutton on the Control Station front panel.

**! WARNING !**

**A FIRE HAZARD EXISTS WHEN CUTTING HIGHLY REFLECTIVE MATERIAL (SUCH AS ALUMINUM, BRASS OR COPPER) AND WHEN USING THE RAPID PIERCE OPTION. DO NOT OPERATE THE MACHINE UNATTENDED WHEN THESE CONDITIONS APPLY.**

If desired, the X, Y, and Z moves in a program can be checked without actually cutting the material by using Program Test mode. Select the "Program Test" button on the Program Modes Control Bar (or press the F2 key on the keyboard). The 'Program Test' dialog then appears:




To test a program with Z-axis motion, press the "Z-axis Up and Down Moves" button, and then select "OK". To run without moving the Z-axis, do not press Z-axis Up and Down Moves. Press "CYCLE START" pushbutton/indicator to start the program. To turn off Program Test mode, press the "Program Test" button again (or press the F2 key). The "Program Test" button will become gray to indicate Program Test mode is off.

*Note: Program execution can be interrupted at any time by performing one of the following actions:*

- *Press the "CYCLE STOP" pushbutton on the Control Station front panel.*
- *Press the "FEEDHOLD" pushbutton on the Remote Station.*

*Note: In an emergency situation, press the "EMERGENCY STOP" pushbutton. This action will halt program execution, shut down the laser high voltage, and stop all drive motion.*

## 7. Check Cutting Performance

While the first few parts in a program are being cut, check the Process Status Bar to verify that the actual assist gas pressure, laser power and feedrate match the programmed values. If desired, stop the machine after the first few parts have been cut to check the part quality. For additional assistance in diagnosing cut quality problems, refer to the **Process Parameters and Diagnostic Guide**.

# ERROR RECOVERY

## CUTTING HEAD BREAKAWAY

The cutting head assembly is magnetically attached to the Z-axis block. This connection limits damage when the cutting head strikes an obstruction. When proximity sensors detect the breakaway connection has opened, the machine is placed in Cycle Stop mode and the system alarm message "Nozzle breakaway detected" is displayed.

To re-attach the cutting head and continue the cutting process, follow this procedure:

1. Except as noted, *do not press* the on-screen "Rewind" button on the Program Modes Control Bar anytime during this procedure.
2. Put the control in Jog mode by pressing the "Jog" button on the Operator Console Control Bar.
3. Retract the Z-axis by pressing the "Z UP" button on the Control Station front panel. The cutting head will be suspended from the breakaway flange by cables or a protective head bellows. If the cutting head can be easily reached from the operator control side of the machine without jogging the X-axis or Y-axis, proceed to Step 5.
4. Verify the "RAPID TRAVERSE" button on the Control Station front panel is not lit. Jog the cutting head within easy reach of the operator control side by pressing the "-Y" pushbutton.
5. Re-attach the cutting head to the magnetic breakaway flange. Concentricity is maintained by a pilot diameter that must be engaged before the flanges will pull together to make the connection. Rotate the head to align the suspension cable support rods vertically.
6. Restore laser high voltage by pressing the "High Voltage" button on the Operator Console Control Bar.
7. Remove the obstruction that caused the head to break away. Use a tool that allows the operator to remain on the floor. If necessary, jog the pallet to the load/unload position for easy access. DO NOT climb onto the machine or pallets to remove an obstruction.

**! ! DANGER ! !**

**DO NOT CLIMB ON MACHINE OR WORK SUPPORT PALLET.**

## RESTARTING A PROGRAM

Follow these steps to resume an interrupted cutting program at the point where the interruption occurred:

1. Re-attach the cutting head if it was dislodged. Restore laser high voltage. See Cutting Head Breakaway.
2. Put the control in Auto mode by pressing the "Auto" button on the Operator Console Control Bar.
3. Activate Trace mode by pressing the "Tracing" button on the Program Modes Control Bar.
4. Press the "CYCLE START" button on the Control Station front panel. The cutting head will return to the position at the beginning of the program block that was being executed when the interruption occurred. If the interrupted block did not contain any motion commands, the cutting head will not move when "CYCLE START" is pressed.
5. Move the cutting nozzle through the program as close to the point of interruption as possible using the "Trace FORWARD" or "Trace REVERSE" buttons on the Control Station front panel. Functions such as **Beam ON**, **Assist Gas ON**, etc. are not executed while in Trace mode. Release the "Trace FORWARD" or "Trace REVERSE" button at any time to stop motion.

*Note: The Z-axis can be jogged up or down as needed while Trace mode is active.*

6. Deactivate the Trace mode by pressing the "Tracing" button again.
7. Resume normal program execution by pressing the "CYCLE START" button.

*Note: DO NOT press "CYCLE STOP" after moving the cutting nozzle in Trace mode. Pressing "CYCLE STOP" (then "CYCLE START") will make the program resume at the beginning of the interrupted block, instead of at the location reached using the "TRACE FORWARD" or "TRACE REVERSE" buttons.*

## BATCH MODE PROGRAM EXECUTION

Batch mode Program Execution occurs when the control executes a loaded batch document. See the **BATCH PROGRAM FILE** topic for a description of batch documents. When running a batch document, the control executes a series of programs including all commands to exchange pallets. In Batch mode, the pallets exchange positions before the first program is executed and after each program ends. The programs in the batch document cannot contain any pallet exchange commands (**M50**s).

*Note: Programs that use Batch mode should process an entire single sheet of material.*

The operator must press "CYCLE START" to begin each program in the batch and press "PALLETS NOT READY" to approve each pallet exchange. The following procedure describes the sequence.

To operate the Laser System in Batch mode:

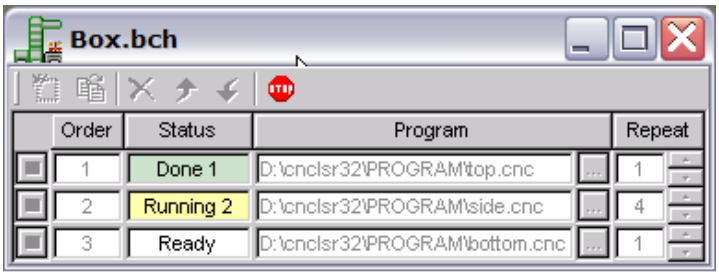
1. Open a batch document.
2. Put the control in Auto mode by pressing the "Auto" button on the Operator Console Control Bar.
3. Select the **File | Load Current Batch** command. Notice that the editing function buttons and text displays in the batch document window are now grayed out and disabled.
4. The *CNC Run Window* should open with the first program loaded.
5. Press the "CYCLE START" button to start the batch.
6. Load the material for the first program on the pallet in the Load/Unload position.
7. Press the "PALLETS NOT READY" button to acknowledge that the material for the first program has been loaded, and the pallets will exchange.
8. When the pallet exchange is completed, the first program will start running.
9. While the first program is running, load the material for the second program on the pallet in the Load/Unload position.
10. When the pallet is ready with the material for the second program, press the (illuminated) "PALLETS NOT READY" button. The button light will extinguish, indicating that the pallets will exchange positions when the control completes the first program.
11. After the pallets exchange positions, the control will load the second program.
12. Press "CYCLE START" to begin executing the second program.
13. While the second program is executing, remove the parts and scrap from the first program and then load the material for the third program.
14. Press the (illuminated) "PALLETS NOT READY" button to allow the pallets to exchange when the second program ends.

15. After the pallets exchange positions, the control will load the third program.

16. Press “CYCLE START” to begin executing the third program.

17. This sequence will continue until the last program in the Batch has been completed and the pallet with those parts is in the Load/Unload position.

When running programs in Batch mode, the control updates the **Status** column in the Batch window to indicate the progress as shown:




# MACHINE SETUP CHECKING

## INTRODUCTION

The **Machine Setup Checking** function is designed to help ensure that a Laser System’s current focusing lens, nozzle tip and assist gas setup meets the requirements of the cutting process used by a part program before it is loaded or executed. Machine Setup Checking is an optional feature that can be enabled or disabled by the machine operator to help prevent scrapped material that can result from running a cutting program when an incorrect lens or nozzle is installed, or when the required assist gas is not available.

## HOW IT WORKS

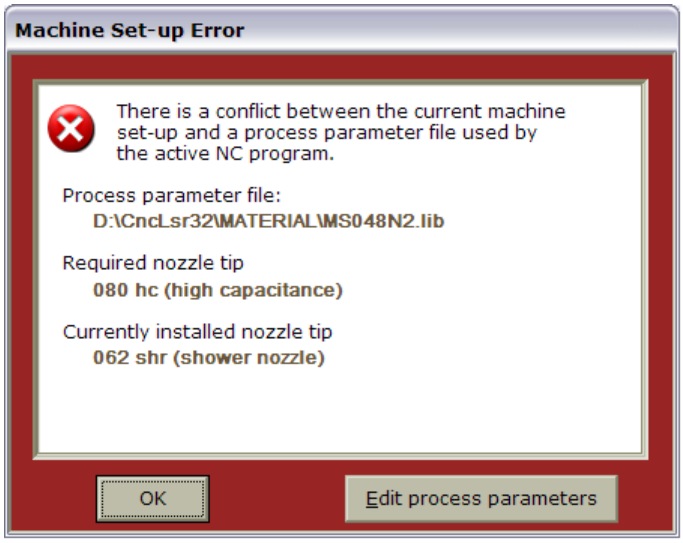
Each Process Parameter file contains parameters for required focusing lens, nozzle tip and pierce and cut assist gas. The operator uses the *Operator Set-up* window to specify which nozzle and lens are currently installed, and how the assist gas ports are configured. Each time a “**G89** P ProcessFile.lib” statement is executed while the CNC is loading or executing a part program, the current cutting head and assist gas settings shown in the Operator Set-up window are compared with the settings from the Process Parameter file indicated by the **G89** statement.

If **Cutting Head Setup Checking** is enabled (see **Cutting Head Setup** page description in the *Operator Set-up* window topic), then both the focusing lens and nozzle tip settings stored in the Process Parameter file must match those currently displayed in the *Cutting Head Setup* window.

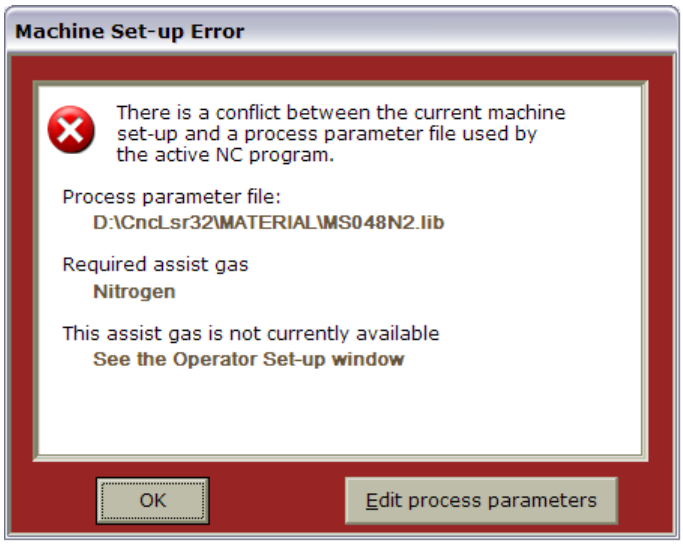
If **Assist Gas Setup Checking** is enabled (see **Z-axis Setup** page description in the *Operator Set-up* window topic), then both the cut and pierce assist gas types stored in the Process Parameter file must be mapped to one of the machine’s assist gas ports, as indicated in the Assist Gas Setup window.

If there are any conflicts between the two sets of parameters, a *Machine Setup Error* window (see figures that follow) will pop up, describing the nature of the setup conflict. As a result of the machine set-up error, the program load or program execution process will be terminated. To resolve a cutting head setup conflict, install the correct nozzle and lens, and/or change the settings in the *Operator Set-up* window to match. To resolve an assist gas conflict, make sure the gas supplies are correctly connected to the distribution manifold, and/or change the assist gas port configuration in the *Operator Set-up* window to match.

If a setup error is reported even though the machine is properly set up and the *Operator Set-up* window shows the correct settings, then the Process Parameter file may contain an incorrect setting. Clicking the “Edit Process Parameters” button in the machine setup error window will open the Process Parameter file that has a setup conflict.

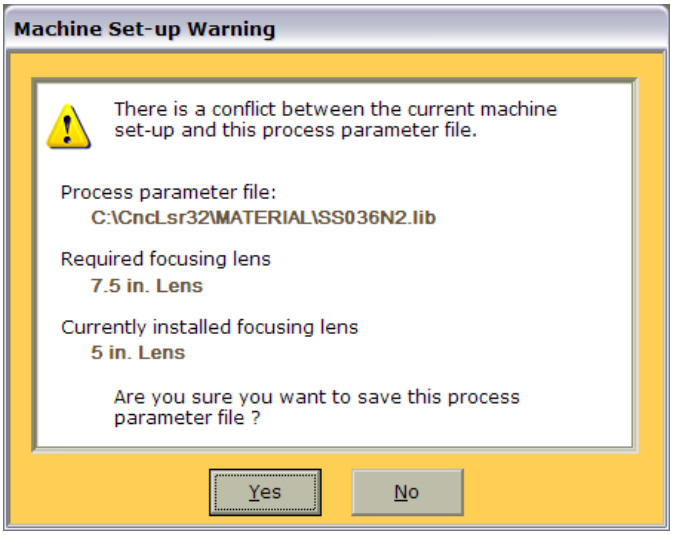

Machine Set-up Error
There is a conflict between the current machine set-up and a process parameter file used by the active NC program.
Process parameter file:
D:\CncLsr32\MATERIAL\MS048N2.lib
Required nozzle tip
080 hc (high capacitance)
Currently installed nozzle tip
062 shr (shower nozzle)
OK
Edit process parameters

***Machine setup error resulting from a nozzle tip conflict.***


Machine Set-up Error
There is a conflict between the current machine set-up and a process parameter file used by the active NC program.
Process parameter file:
D:\CncLsr32\MATERIAL\MS048N2.lib
Required assist gas
Nitrogen
This assist gas is not currently available
See the Operator Set-up window
OK
Edit process parameters

***Machine setup error resulting from an assist gas conflict.***

A Machine Setup Warning (see figure) will be reported if a Process Parameter file is edited to cause a machine setup conflict after that file has been loaded via a **G89** statement by the currently executing part program. Although a machine setup warning will not cause the running program to stop immediately, if the file containing a conflict is saved anyway, by clicking the "Yes" button in the setup warning window, then the next time **G84** is executed, a machine setup error will cause program execution to stop. (Exception: If the part program uses more than one Process Parameter file, and the file with the conflict is not the active Process Parameter file when it is saved, a machine setup error will not be reported until the next **G89** command attempts to load that Process Parameter file to make it active.)


Machine Set-up Warning
There is a conflict between the current machine set-up and this process parameter file.
Process parameter file:
C:\CncLsr32\MATERIAL\SS036N2.lib
Required focusing lens
7.5 in. Lens
Currently installed focusing lens
5 in. Lens
Are you sure you want to save this process parameter file ?
Yes
No

***Machine setup warning.***

# AUTO FOCUS CUTTING HEAD

The Auto Focus Cutting Head improves process repeatability by eliminating a manual focus adjustment, and enables process control functions that are not possible with manual focus cutting heads. For example, the program can specify different focus positions for piercing and cutting, and either a fixed focus position or a range of positions based on laser beam length. With automatic focus control, the laser control can automatically change the focus position as X- and Y-axis motion changes the beam length.

## FOCUS CONTROL SYSTEM CONFIGURATION

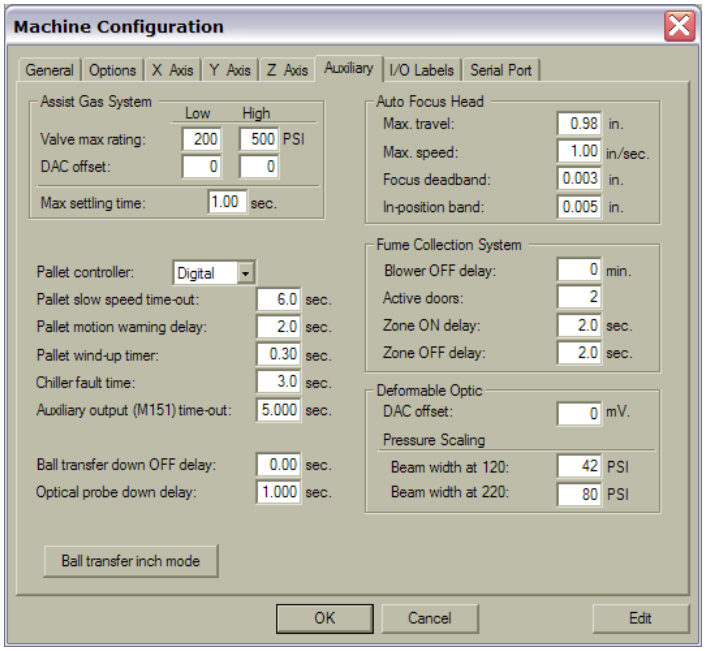
The Auxiliary tab of the *Machine Configuration* window has an Auto Focus Head section containing the following parameters:

*Max. travel:* Specifies the available travel distance for the drive mechanism. The control software uses this parameter as a reference, to prevent the operator from requesting a focus position that exceeds the travel range of the mechanism.

*Max. speed:* Specifies the maximum speed of the Auto Focus drive mechanism.

*Focus deadband:* Specifies the minimum change in the desired focus that causes the drive mechanism to respond. The intended function of this parameter is to prevent unnecessary and trivial movement of the lens holder during dynamic focus adjustment. For example, the Auto Focus control should not move the lens while cutting a small circle, even if the beam length changes enough for the control algorithm to calculate a slightly different lens position.

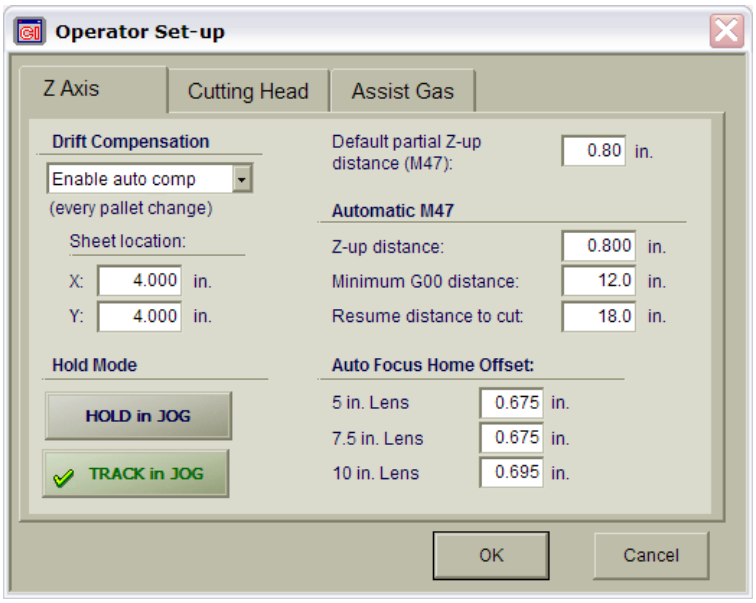
*In-position band:* Specifies the maximum following error in the Auto Focus drive that the control will tolerate when starting a cut sequence. The "following error" is the difference between the actual lens position and the lens position required for the desired focus. If the "following error" is greater than the In-position band, the (**G84**) Pierce/Cut operation will wait until the Auto Focus drive reduces the following error before transferring control to the next program block. The In-position band is used to compensate for cases when the X- and Y-axis drives change the beam length faster than the Auto Focus drive can maintain the corresponding lens position. That condition can occur during high-speed cutting.



# AUTO FOCUS SETUP

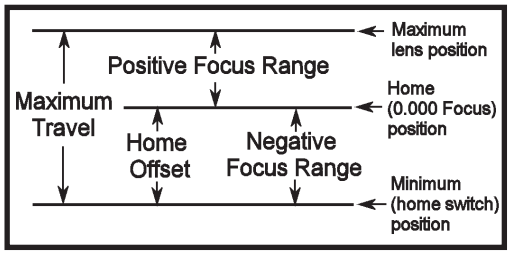
## FOCUS CONTROL SYSTEM HOME OFFSETS

The *Z-axis Setup* window has a section for *Auto Focus Home Offset* settings. These fields specify the home offset distance for the Auto Focus drive mechanism. To accommodate lens variations, there is a setting for each lens drawer. When running a program, a **G89** Process Parameter specifies the active lens drawer. Refer to **FOCUS CALIBRATION - Auto FOCUS HEAD** in the **CALIBRATION AND ADJUSTMENTS** section for the procedure used to set the *Auto Focus Home Offsets*.



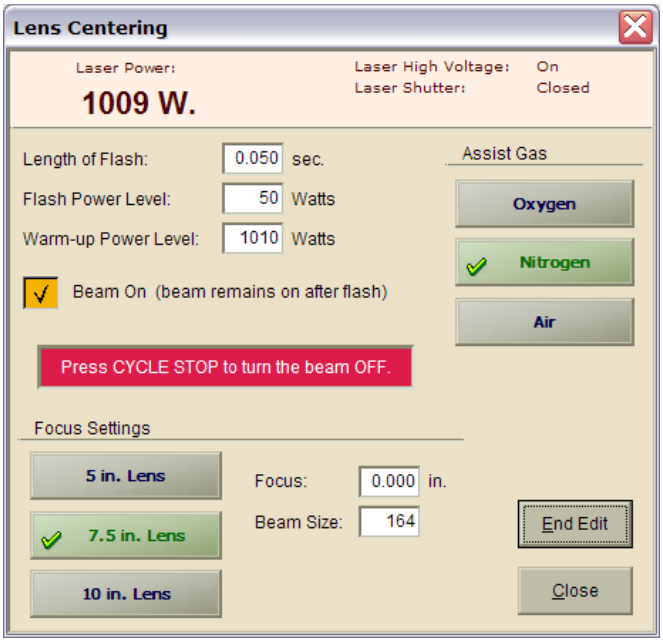

## FOCUS SETUP FOR LENS CENTERING

In the *Lens Centering* window, the user can specify which lens drawer is installed and what focus setting to use when flashing the beam for tape shot. The Focus setting affects the "spot size" of the tape shot. The allowable range for focus setting values depends on the *Auto Focus Max. Travel* and *Home Offset* settings (see Focus Range diagram). A set of Lens buttons is used to specify the active lens drawer so the control knows what home offset to apply to the focus setting.




***Focus Range***

When the operator selects the "End Edit" button (and the focus value is inside the allowable range) the drive mechanism immediately moves the lens holder to the new setting. The operator can then execute a tape shot using the specified focus setting. The *Focus* setting in the *Lens Centering* window is temporary. When the Lens Centering window closes, the drive mechanism returns the lens holder to the focus setting of the most recent Process Parameter file called by **G89**, or specified explicitly with **G102**.




### LENS CENTERING ADJUSTMENT

The front of each lens drawer has two lens centering screws with knurled knobs (see Auto Focus Cutting Head photograph in the **LENS CENTERING PROCEDURE** topic). Open the Auto Focus access door to adjust the lens centering screws and then close the door to take another tape shot.

## FOCUS-RELATED PROCESS PARAMETERS

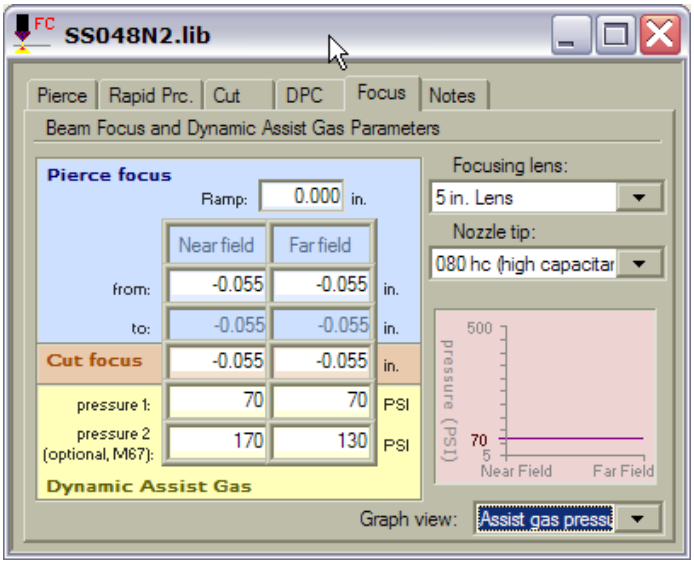
During program execution, automatic focus settings are specified in Process Parameter files, or through **G102** command line arguments.

*Note: CINCINNATI provides Process Parameter files with Focus settings based on zero focus calibrated in the Far field of the cutting area.*

The Focus page of the *Process Parameter* window contains the Near field and Far field settings for focus position and dynamic assist gas pressure used for automatic operation:

*Ramp (Pierce focus):* This setting configures the pierce process to change focus position during the pierce sequence. The pierce sequence begins with a pierce focus position based on the *Pierce from* settings specified for the near and Far field. During the pierce time, the focus position changes by the distance specified with the *Pierce Ramp* value, ending with a focus position based on the near and Far field *Pierce to* settings.

*Near field focus (Pierce and Cut):* These settings specify the position of the focal point for piercing and cutting when the nozzle is in the Near field of the laser beam. The Near field is where the cutting head is closest to the laser resonator. The focus settings specify positions relative to the 0.000 inch focus position. A negative value specifies a focus position below the nozzle tip. The allowable range for focus setting values depends on the *Auto Focus Max. Travel* and *Home Offset* settings (see Focus Range diagram ).




*Far field focus (Pierce and Cut):* These settings specify the focal position for piercing and cutting when the nozzle is in the Far field of the laser beam. The Far field is where the cutting head is farthest from the laser resonator. The focus settings specify positions relative to the 0.000 inch focus position. A negative value specifies a focus position below the nozzle tip. The allowable range for focus setting values depends on the *Auto Focus Max. Travel* and *Home Offset* settings (see Focus Range diagram ).

*Focusing Lens:* This setting specifies which lens drawer will be used. This setting tells the Auto Focus control software to apply the *Home Offset* value for the appropriate lens.

# AUTO FOCUS OPERATION

## HOMING THE FOCUS CONTROL SYSTEM

Before the Auto Focus can be used in a program, the focus control system must be “homed”. If Auto Focus homing is required, simply pressing the “RESET” button on the Control Station front panel when the control is in Jog mode will start the process. The Auto Focus drive motor will then move the lens holder down to a fixed reference position (until it encounters the home switch) and then move upward the distance specified by the “Auto Focus Home Offset” field in the *Operator Set-up* window. The homing operation ends with the lens at the 0.000 inch focal position.

### PROGRAMMING

A program can specify focus settings from a Process Parameter file using **G89**, or explicitly using **G102**. The **G102** macro call specifies focus settings with these arguments:

- **Q** - Pierce focus , Near field
- **R** - Pierce focus , Far field
- **U** - Cut focus , Near field
- **V** - Cut focus, Far field

**G102** interprets the specified focus settings in the active units mode (**G20** inch, **G21** mm). The focus settings specify the lens position relative to the zero or “home” position. The Auto Focus control applies the **G102** focus settings to the Home Offset parameter of the lens last specified by a **G89** Process Parameter file.

## CHANGING FOCUS WITH BEAM PATH LENGTH

The Process Parameter settings for Focus specify the lens position for running a program. If the Near field and Far field focus settings are different, then Auto Focus changes the lens position as the beam path length changes. As the X- and Y-axes change the beam length, the software calculates a desired focus position between the Near field and Far field settings. The graph on the Process Parameter Focus page shows the linear relationship between beam path length and focus position. The actual focus setting is displayed in real time on the Process Status Bar.

The Auto Focus drive mechanism only moves the lens on a periodic basis (not continually). When the control software determines that the difference between the desired lens position and the actual lens position is more than the Focus Deadband **(**see Focus Control System Configuration**)**, the control moves the lens. By making only periodic adjustments, the control avoids excessive cycling of the drive mechanism and minimizes process delays.

## STANDOFF CONTROL

The Laser System controls Nozzle Standoff position and Lens Focus position independently. However, the position of the beam focus relative to the material surface changes when standoff changes. The Focus set point only specifies a lens position relative to the home position (in other words, relative to the nozzle). A new standoff setting may require a different focus setting to achieve acceptable results. When editing a Process Parameter file (or an explicit **G102** call), the user is responsible for maintaining an appropriate combination of standoff and focus settings.

# WINDOWS ADMINISTRATION

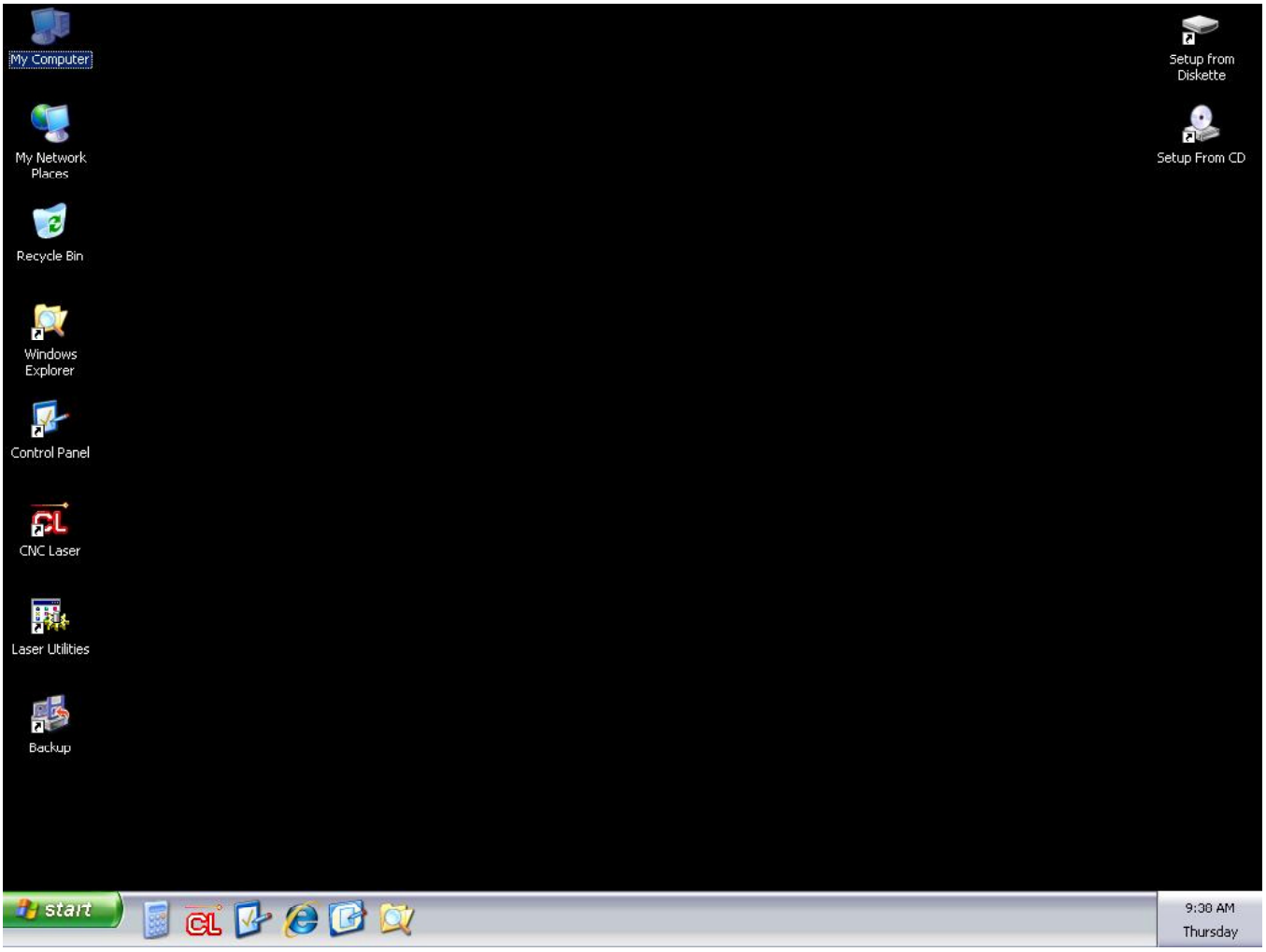
A number of Windows administration procedures are available, including: machine backups, CINCINNATI INCORPORATED software upgrades, creating emergency repair disks, and setting system date and time. All of these functions are available in the Setup or Manager user accounts.

The Setup account has administrator privileges and can be used for network setup. The applications available on the Setup desktop are limited to those administrative tasks allowed to the customer. The Laser System cannot be operated from the Setup account. The Manager account has access to all applications available to the Operator account, plus the ability to copy and delete files, back up data, and install programs using the Setup account user name and password (Windows XP Only).

To log in to the Setup account, first log off of the current user. This will open the main login screen. Press the Ctrl+Alt+Delete key combination twice to bring up the classic login box. Type in “setup” for the user name and then the password to log in.

*Note: If connected to a domain, the classic login box will already be displayed upon log off. (Windows XP Only).*

## MANAGER ACCOUNT




***The Windows desktop as it appears for the Manager account***

*My Computer*, *My Network Places*, *Recycle Bin*, and *Windows Explorer*: Typical shortcuts for Windows.

*CNC Laser:* Starts the Laser System CNC/HMI program.

*Start Button:* Use this to access other available applications.

*Time/Date:* Double-click the clock in the lower right-hand corner of the screen to modify the time or date.

*Backup Folder:* Shortcuts to backing up the Laser control along with Help tutorial.

## LASER UTILITIES FOLDER

*Process Parameters and Diagnostic Guide:* Opens the document to view Process Parameters.

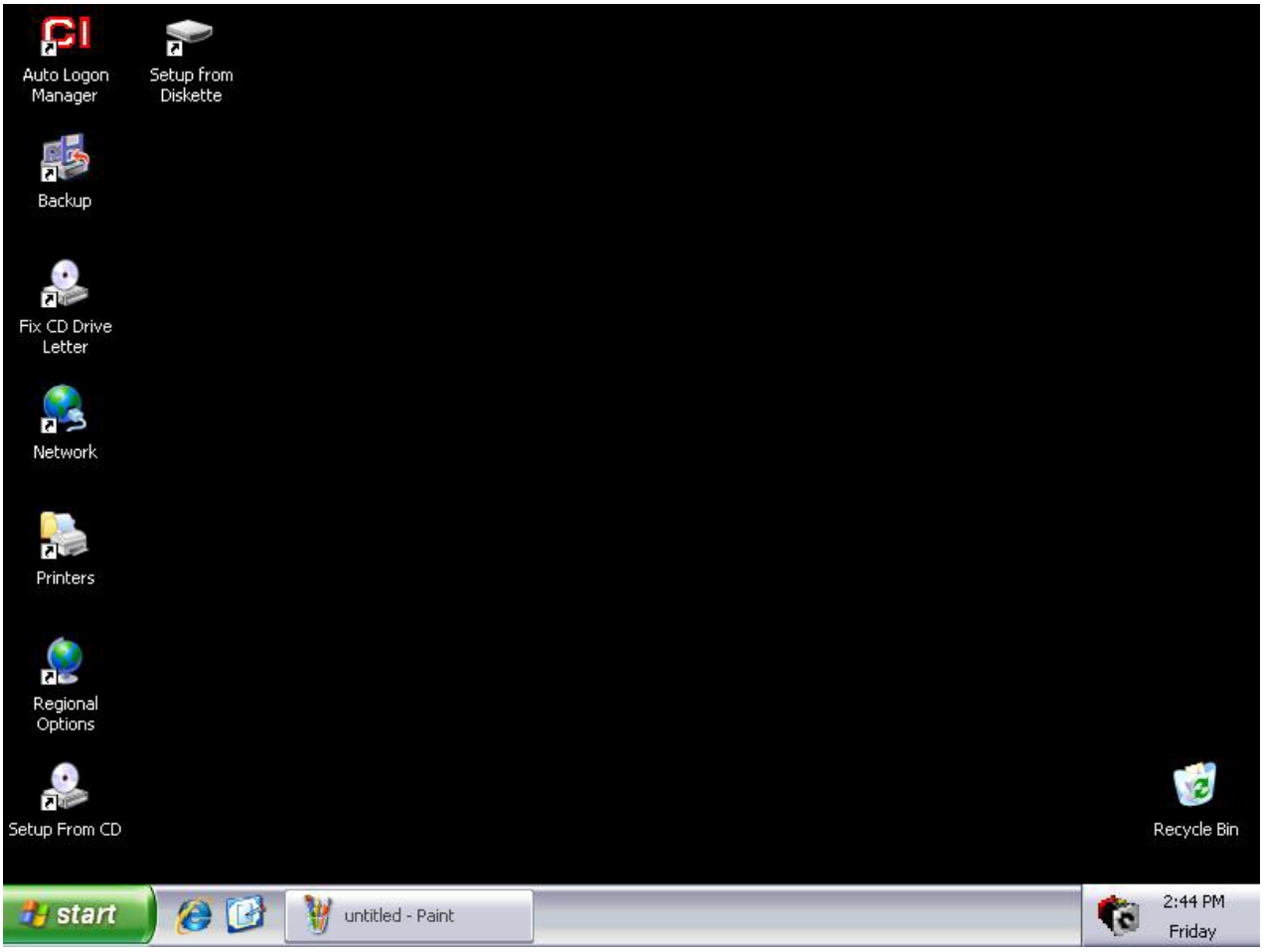
*Frequency Checker:* This applet checks if any Process Parameter files in the selected folder command laser resonator pulse frequencies outside a specified range. It is intended for use only by CINCINNATI INCORPORATED Laser Service representatives.

## SOFTWARE INSTALLATION AND UPGRADES

On Laser Systems with Windows XP controls, the Manager user can install or upgrade control software.

Close the Laser System **CNC/HMI** application. Insert the floppy disk or CD, if available, from CINCINNATI INCORPORATED then browse to the CD or disk using My Computer or Explorer and click on the "setup.exe" file. The software may need Administrative rights to run, and a login dialog box may open. Type in the Setup account's user name and password and click "OK". Follow the prompts provided by the Setup software. Files are loaded for the new Laser System **CNC/HMI** application and PLC's. When the **CNC/HMI** application is restarted, the software will automatically perform "PLC Update", which loads the PLC data into the motion controller flash memory. When the PLC update is complete, the **CNC/HMI** application will automatically close, and the upgrade is complete.

## SETUP ACCOUNT




***The Windows desktop as it appears for the Setup account***

*Setup Network:* This is the 'Network' icon from the Control Panel. Use it to install network adapters, protocols and services. See "Network Setup".

*Auto Logon Manager:* Using this applet, the Setup user can configure the Laser System control to automatically logon to a specified account (domain, user name and password) when the system is booted. Automatic logon allows the control to start without requiring the operator to type a user name or password. Auto Logon Manager does not allow automatic logon to the Administrator account.

*Event Viewer and System Information:* These utilities are used to gather diagnostic information about the system.

*Network:* This is the Network applet from the Windows Control Panel. Use it to install network protocols and services, as well as joining an NT domain. See **Network Setup** topic.

*Printers:* This is the Printers applet from the Windows Control Panel. Use it to configure printers that the Laser System will use.

*Regional Options:* This is the Regional and Language Options applet from Windows Control Panel. Use it to configure the display of different languages, time, and other regional settings.

*Setup from CD:* (If CD available) This shortcut allows laser software to be upgraded using applications developed by CINCINNATI INCORPORATED. Insert the software upgrade medium and double-click the appropriate shortcut.

*Note: Laser control may not have a CD drive. Software can be upgraded via USB stick or over the network from the Manager account. Refer to "Software Installation and Upgrades".*

*Time/Date:* Double-click the clock in the lower right-hand corner of the screen to modify the time or date.

*Start button:* Use this to access the touchscreen calibration, backup utility (also used to create an emergency repair disk), and other applications.

## SOFTWARE INSTALLATION AND UPGRADES

### Windows 2000 or XP (If CD or floppy drive available)

Insert the floppy disk / CD from CINCINNATI INCORPORATED and double-click on the "Setup from Diskette" or "Setup from CD" shortcut. Follow the prompts provided by the Setup software. Files are loaded for the new Laser System **CNC/HMI** application and PLC's. When the **CNC/HMI** application is restarted, the software will automatically perform "PLC Update", which loads the PLC data into the motion controller flash memory. When the PLC update is complete, the **CNC/HMI** application will automatically close, and the upgrade is complete.

***IMPORTANT: Do not attempt to install desktop applications without first consulting CINCINNATI INCORPORATED.***

## EMERGENCY REPAIR DISK

A CD or bootable USB stick is included with the Laser control. These will boot the control to an interface allowing recovery of a previously saved hard drive image.

## NETWORK SETUP

The Laser System control incorporates all networking capability offered by Windows, including a diverse array of networking options and properties. Instead of presenting detailed procedures for dealing with all of them, this section describes details that are unique to the Laser System control. The user should contact their company's system administrator for custom network setup.

## JOINING A DOMAIN

To add the laser control to a Windows domain, follow these steps: (Must be logged in as "setup".)

1. Double-click the 'Network' icon on the desktop.
2. In the *Network and Dial-up Connections* window menu, select "Advanced" and "Network Identification".
3. In the *System Properties* window, select the 'Network Identification' tab, and then select the "Properties" button.
4. In the *Identification Changes* window, select the "Domain" option button and enter the domain name in the edit field.
5. Select the "OK" button.
6. When prompted, enter user name and password of a Domain Administrator.

# DRIVE DESIGNATIONS

A: This is the 3.5" Floppy Disk drive, if available.

C: This is the hard drive partition, which contains Windows XP or 2000, and all control applications and data for running the Laser System.

D: This is the hard drive partition, which contains drivers, manuals and other utilities.

F: Flash Drive or USB stick usually labeled "backup". This is used to store the factory image of the laser control and do weekly backups of the laser control files (Windows XP Only). Used for Backup and Restore Wizard on Windows 2000 systems.

Z: This is the CD drive, if available.

### MAPPING NETWORK DRIVES

All Operator and Manager-level accounts have access to Windows Explorer to map a network drive. Following are the basic steps:

1. Double-click 'Windows Explorer' icon on the desktop.
2. In the *Explorer* window, pick the *Tools* menu and select "Map Network Drive".
3. Select the 'Reconnect at logon' check box if desired.
4. Enter the desired drive letter and network folder.
5. To log on to the host as a user other than current account, select "Connect using a different user name" and enter the information.
6. Select the "Finish" button to complete the drive mapping operation.

## BACKING UP THE CONTROL

Controls have pre-installed backup software called **Acronis**. A folder on the Manager and Setup desktops called "Backup" contains all the shortcuts necessary to backup the machine. Some controls are equipped with a 2GB flash drive setup for weekly backups of the control. Newer controls have a 4GB to 8GB drive that contains a factory shipped image as well as a weekly backup of the control. There is a tutorial for using this software and the included shortcuts located in the same backup folder.

## MULTIPLE OPERATOR USER ACCOUNTS

If the laser control is part of a domain, then any authenticated user in that domain is viewed by the control as an Operator. The default user profile is that of the Operator account, so any member of the domain has the permissions, policy restrictions and desktop of the Operator. Network drives are mapped on a per-user basis, so each user selects an individual network drive-mapping configuration.

## TOUCHSCREEN CALIBRATION

If the touchscreen has never been calibrated, or if the current user finds that touching the screen does not move the pointer arrow to the expected location, the user should re-calibrate the touchscreen.

*Note: The touchscreen does not need calibration each time the laser control application starts. However, users of significantly different height or operating position may wish to re-calibrate the touchscreen. Re-calibration can be done at any time. Calibration shortcuts are available in the Manager and Setup accounts and on some controls in the Operator account.*

To begin the calibration procedure, press the Windows "Start" button (or press the "Windows" button on the keyboard) to display the *Start* menu. Choose **Accessories | Calibrate Touchscreen** which will start the calibration routine.

The actual calibration routine may vary, but the general procedure is to press several targets on the screen as directed, and to accept the settings. The procedure adjusts the touchscreen response for the height and operating position of the user.

# ALARMS AND MESSAGES

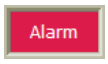
Four types of messages are listed in the tables shown on the following pages:

*Operator Messages:* Operator messages inform the operator when specific conditions affect machine operation, or when specific service or maintenance procedures are required. Operator messages are displayed in either the *Alarms and Messages* window, or in temporary pop-up message boxes.

The *Alarms and Messages* window can be opened quickly by clicking the

indicator on the Status Indicators Control Bar.

*System Alarms:* System alarms indicate critical conditions that inhibit Laser System operation. In the table, descriptions for system alarms that indicate a malfunction begin with the label: “Failure”. System alarm descriptions that begin with the label “Normal” indicate conditions that can occur when the Laser System is functioning normally (for example, “Emergency Stop pressed”). System alarm messages are displayed in the *Alarms and Messages* window.

The *Alarms and Messages* window can be opened quickly by clicking the 

indicator on the Status Indicators Control Bar.

*Laser Alarms:* These are alarm conditions pertaining specifically to the laser resonator or resonator control. For the description of most laser alarms, the table refers to the resonator manufacturer’s manual. Laser alarm messages are displayed in the *Alarms and Messages* window.

*Program Errors:* These messages are displayed in a pop-up message box when a program is loading or executing. The message box closes when the operator selects the “OK” button.

## OPERATOR FYI MESSAGES

| DISPLAYED MESSAGE | DESCRIPTION | POSSIBLE CAUSE(S) |
|---|---|---|
| Antidive is active. | The machine logic has suspended all Z-axis movement due to a lack of material directly beneath the nozzle. (Note: This feature can be disabled with the M130 M code. See Programming Manual). | • The nozzle has passed over a hole.<br>• The nozzle has passed over a slug.<br>• The material is “fluttering” / bouncing (use M130 to remedy).<br>• The X/Y feedrate is too great and/or the material is warped (use M130 to remedy). |
| Autofocus loss-of-counts detected. | The control monitors the number of encoder counts between the power-up position and the home position. If the distance corresponding to that number changes by more than 0.015" (0.38 mm) on consecutive homing attempts, this message is displayed. | • Encoder connection problem.<br>• Ball screw-motor alignment changed.<br>• Loose drive belt inside head. |
| Beam purge loss of pressure alarm disabled. | Beam Purge Fault Override parameter has been enabled in machine configuration / Options. Machine will operate without purge gas (for use by CINCINNATI Service personnel only). | |
| Cannot trace though pallet moves. | Trace mode will not execute pallet moves, as it is intended as a recovery tool for the present job/pallet. | |
| Chiller is not turned on. | Chiller main contactor is not on and is not being commanded to turn on. | • If chiller is running (switch set to LOCAL) and this message appears:<br>• Bad wire connection from I/O PC board to chiller contactor. |
| CYCLE STOP pushbutton pressed. | One or more of the “CYCLE STOP” or “FEEDHOLD” buttons are pressed. | • If buttons are not being pressed:<br>• Bad wire connection to button.<br>• Ball Transfer option turned on without installing Ball Transfer remote control. |

| DISPLAYED MESSAGE | DESCRIPTION | POSSIBLE CAUSE(S) |
|---|---|---|
| Deformable Optic not enabled. | Deformable optic pump does not enable until reset is pressed with drives on. | • Drives were turned on, but RESET not pressed yet. |
| Deformable Optic out of tolerance. | Deformable optic pump is moving to a new pressure setting, and is temporarily out of tolerance. | |
| End of history buffer or beginning of program reached. Reverse Tracing is disabled. | Tracing backwards beyond start of program is not possible. In addition, a fixed number of executed program blocks are held in memory while a program is running. This memory area, known as the “history buffer”, limits how far a program can be traced in reverse. | • Large programs contain more blocks than the history buffer can hold. Attempts to trace backwards beyond the last block in the history buffer will cause this alarm. |
| End of program reached. Forward tracing is disabled. | Tracing operation has reached the last line of the program. Press “Trace REVERSE” to back up to desired location for resumption of cut. | |
| Feedrate override is set to 0%. | The Feedrate Override setting is set to 0%, which suspends all X/Y movement. | |
| Gantry is out of square – Beam flash is disabled. | Cut area enclosure gate was opened, actuating the gantry clamp; however, X1 encoder position does not match X2 encoder position. Beam is disabled. | • Abnormal load on gantry<br>• Personnel are leaning/pushing against gantry when gate is open. |
| HV safety interlock is bypassed. | The “HIGH VOLTAGE” interlock switch on the control station side panel is set to BYPASS. | |
| Laser beam is disabled while setup parameters are being edited. | No Lens Centering or Mirror Alignment beam flash is permitted until editing has been completed. | |
| Laser high voltage is off. | High voltage disabled by interlock circuit in laser resonator or CINCINNATI control. | If no interlocks tripped:<br>• Bad connection from High Voltage relay contact to I/O PC board.<br>• Bad connection / relay in resonator enclosure. |
| Lower ball transfer to move pallets. | The Ball Transfer option is enabled **and** a pallet move was attempted with the ball transfer raised. | If ball transfer are down:<br>• Ball Transfer Down limit switch out of adjustment.<br>• Bad wire connection to Down switch. |
| Lower Pallet Special Function mode (M43) is active. | The Lower Pallet Special Function option is enabled, and a program commanded M43. (See Programming Manual.) | |
| Material Handling System minor fault. | The material handling system has faulted. Laser program execution can continue. | • The material handling system has an operational fault. See Material Handling System manual. |
| Noncontact Head Installed prox. switch failed. | The noncontact head proximity switch opened after a program started with the switch closed. The program continues unless a breakaway switch also opens. The control displays this message as a System Alarm at the next program interruption. | • Proximity sensor mounted too far from flange.<br>• Faulty switch.<br>• Faulty cable. |
| Noncontact tip touch warning – check tip alignment. | A brief tip touch indication is not enough to trigger an alarm, but can be an indicator of a loose or problematic tip. Per the settings in the Machine Configuration, these brief tip touches can be reported to the operator, in order to anticipate cutting problems due to a loose tip, etc. | • Loose tip.<br>• Material is vibrating during the cut process, due to assist gas. |
| Pallet loading in progress – pallets are disabled. | Automated Pallet Loading option has been enabled, and the pallet loader is sending the “loading” signal. | |
| Positioning nozzle for standoff calibration. | A Standoff Calibration or Drift Compensation move is in progress. This message will only be displayed while the noncontact head is moving down to contact the material, and then reversing to the desired standoff setting. | |
| Remote station is enabled - no axis or pallet moves permitted from console. | Remote pendant station key selector is set to ENABLED. | |

| DISPLAYED MESSAGE | DESCRIPTION | POSSIBLE CAUSE(S) |
|---|---|---|
| Safety gate bypassed by Remote Station enable switch. | Enabling the Remote Station allows operator to open gate and perform lens centering or mirror alignment. Drive power will stay on in this case, allowing Z and Auto focus movement. | |
| Safety zones bypassed by Remote Station enable switch | Enabling the Remote Station overrides the normal safety device interlock mechanism and allows operator to perform lens centering or mirror alignment. Drive power will stay on in this case, allowing Z and Auto focus movement | |
| Shutter is locked. | The SHUTTER selector switch on the control station side panel is set to LOCK/CLOSE. | • If SHUTTER switch is not set to “LOCK/CLOSE”:<br>• Bad connection from console assembly to I/O PC board. |
| Shutter test active - reset when done. | The shutter mechanism was actuated by pressing the “SHUTTER” button on the Operator Console Control Bar with High Voltage off. | |
| Standoff drift compensation was not done on this pallet. | Standoff Calibration CHECK is not valid since offset was calculated on the other pallet. Must either select RECALIBRATE or switch pallets. | |
| X1 encoder loss-of-counts detected. | The control monitors the number of encoder counts between the power-up position and the home position. If the distance corresponding to that number changes by more than 0.015" (0.38 mm) on consecutive homing attempts, this message is displayed. | • Encoder connection problem.<br>• Ball screw-motor alignment changed.<br>• Defective encoder. |
| X2 encoder loss-of-counts detected. | See “X1 encoder loss-of-counts detected”. | |
| Y encoder loss-of-counts detected. | See “X1 encoder loss-of-counts detected”. | |
| Z-axis antidive is disabled. | The **Antidive** feature has been disabled by a program commanding M130.<br>See **LASER NC PROGRAMMING** topic. | |

# SYSTEM ALARMS

| DISPLAYED MESSAGE | DESCRIPTION | POSSIBLE CAUSE(S) |
|---|---|---|
| Assist gas valve failed to maintain target pressure. | [Normal] Assist gas servo valve “in range” signal stayed low for too long. (Time-out period adjustable on Configuration screen.) | • Out of gas / supply turned off.<br>• Commanded pressure too high. Supply pressure should be 50-100 PSI (345 to 690 kPa) more than required pressure.<br>• Bad wire connection from valve to console I/O PC Board.<br>• Faulty/clogged assist valve. |
| Auto Focus amplifier fault. | [Failure] The Auto Focus motor/drive is indicating a fault. For more description, see “Auto Focus Cutting Head” in this section. | • The motor bottomed out due to incorrect home offset or max travel settings.<br>• A failure occurred in the drive mechanism.<br>• Fault indicated by drive LED in I/O enclosure. |
| Auto Focus Head connection fault. | [Failure] The Auto Focus Head cable is not connected. For more description, see “Auto Focus Cutting Head” in this section. | • Intermittent connection.<br>• The cutting head was changed. |
| Auto Focus Head door is open. | [Normal] Auto Focus Head door must be closed before cutting can begin. | • The door was opened intentionally.<br>• The door switch failed. |
| Auto Focus Head not homed. | [Normal] The Auto Focus head has not been homed, or failed to complete the homing operation. | • Drive power turned off.<br>• Cutting head recently installed. |
| Axes have not been homed. | [Normal] X-axis and Y-axis motors must be “phased” to the magnet track and homed before continuing. | |
| Auxiliary input (M151) did not turn on. | M151 was active beyond a prescribed time-out period, indicating M151 input did not cycle on and then off. | • M151 output relay malfunction.<br>• External device did not actuate M151 input on and then off.<br>• M151 input malfunction. |
| Axes must be ACTIVE to align the beam. | [Normal] X1 and X2 motors must be enabled so the beam optics will be aligned properly. | |
| Axis amplifiers are not enabled - press RESET to re-enable. | [Normal] The “RESET” button must be pressed to reactivate the X1, X2 and Y-amplifiers, following a loss of power to the DRIVES. (DRIVES power must be re-established before this message will appear.) | • The DRIVES selector switch was turned to “LOCK”.<br>• An Auto Shutdown occurred, which automatically causes DRIVES power to turn off.<br>• An X, Y or chiller fault occurred, which automatically causes DRIVES power to turn off. |
| Ball transfer request (M90/M91) timed out. | [Failure] M90 or M91 was active beyond a prescribed time-out period, indicating ball transfer limit switch input did not turn on (if M91), or off (if M90). | • Loss of electrical power to motor.<br>• Ball transfer limit switch malfunction. |
| Beam command with head up. | [Failure] The Z-axis was retracted when either the beam was commanded to be on or the shutter was commanded to be OPEN. | • Z-axis encoder malfunctioned.<br>• Z-axis encoder cable defective. |
| Beam purge pressure fault. | [Normal] The beam purge pressure switch opened for more than 500 milliseconds, indicating that less than 8 PSI (55 kPa) exists at the sensor in the purge gas supply. (Note: This alarm will not appear if “Beam Purge Fault Override” service code is set.) | • Beam purge gas supply is low or turned off.<br>• Faulty pressure switch.<br>• Power failure. |
| Chiller fault. | [Failure] After commanded to start, main contactor inside chiller did not energize before fault timer elapsed. Chiller fault timer is an Auxiliary Configuration parameter. | • Chiller selector switch set to STANDBY.<br>• Blown fuse/overload tripped in chiller.<br>• Bad wire connection from chiller to I/O PC Board.<br>• Power failure.<br>• Chiller requires more time to start than fault timer allows. |
| Chiller temperature out-of-range. | [Failure] Chiller unable to maintain temperature within 5 degrees of setpoint. | • Filters dirty/clogged.<br>• Side panel not installed on chiller.<br>• 1 or more fans not running. |
| CNC watchdog failure. | [Failure] PC and motion controller experienced a break in communications. | |

| DISPLAYED MESSAGE | DESCRIPTION | POSSIBLE CAUSE(S) |
|---|---|---|
| Console Input board failure. | [Failure] The console I/O board is no longer communicating with the CNC control. (If this message is displayed, ignore all other System Alarms until this message is cleared.) | • 24Vdc power supply failure.<br>• Faulty connection in flat 10-conductor I/O board cable (console or PC enclosure).<br>• Faulty connection at I/O communications board (PC enclosure).<br>• Faulty connection in ribbon cable from motion controller to communications board (PC enclosure). |
| Console Output board failure. | [Failure] See “Console input board failure “. | See “Console input board failure”. |
| Console warning zone has been entered | [Normal] An external proximity safety device (safety mat or area scanner) is indicating that an operator warning zone has been entered. | If safety device is not visibly tripped:<br>• Bad wire connection from device controller to I/O PC board.<br>• Faulty CR1 relay or timer relay.<br>• Faulty safety device / controller.<br>• Power failure |
| Deformable optic pump fault. | [Failure] The deformable optic circuit was unable to reach its target pressure within 6 seconds. | • Coolant lines / orifice are plugged.<br>• Faulty connection in pump or pressure transducer. |
| Emergency stop pressed. | [Normal] One of two “E-STOP” buttons has been pressed. High Voltage, X/Y/Z motion and pallets are disabled. | If “E-STOP” button not pressed:<br>• Faulty wire or relay connection on console or load frame “E-STOP” button.<br>• Power failure. |
| Fume blower fault. | [Failure] 3-phase motor starter contact did not close within 3 seconds of being energized. | • Over current dial set too low on motor starter—see schematics for proper setting.<br>• Motor failure / over current.<br>• Starter circuit breaker left in off position.<br>• If starter circuit breaker is not trip:<br>• Bad wire connection between motor starter and I/O PC Board.<br>Or<br>• Wrong size starter for selected fume collection unit. |
| High voltage must be on to cut, align the beam or calibrate. | [Normal] An operation was attempted that requires High Voltage to be on. | |
| HSS electronics fault. | [Failure] The HSS (Height Sensing System) electronics indicated a fault on the digital line. | |
| HV Interlock must NOT be BYPASSED to align the beam or run a program. | [Normal] The “High Voltage” Interlock switch on the console is set to BYPASS. | |
| Internal PMAC error: Circle radius. | [Failure] Part programming error. Ending point specified for arc/circle cannot be reached using specified radius. | |
| Internal PMAC error: Run-time. | [Failure] Motion controller run-time error. | |
| Internal PMAC error: Servo. | [Failure] Motion controller servo error. | |
| Lens coolant failure. | [Normal] Less than 0.1 GPM flow through lens cooling circuit. | • Chiller switch set to STANDBY.<br>• Chiller disconnect set to “OFF”. |
| Lens damage detected – inspect focusing lens. | [Normal] The optical sensor detected smoke or debris near the cutting lens. | • Contamination in the beam delivery system.<br>• Fumes from debris on lens top surface. |
| Lower ball transfer to move pallets | [Normal] Reminder that the ball transfer mechanism must be lowered before a pallet move request can be executed. | If ball transfer are down and message persists:<br>• Check for loose wire.<br>• Verify prox switch is in range. |
| Lower pallet must be in the cutting position in order to execute M43. | [Normal] Lower Pallet Special Function can only be invoked when the lower pallet is IN.<br>See **LASER NC PROGRAMMING** topic. | |
| Main drives are off. | [Normal] Master relay that supplies power to X/Y/Z and pallet drives is off. High voltage is disabled. | If drives were not manually turned off:<br>• **Auto-Shutdown** feature was recently selected.<br>• An X, Y or chiller fault occurred, which automatically causes drive power to turn off.<br>• Power failure. |
| Material clamps must be closed before moving pallets. | [Normal] Close the laser pallet power clamps before jogging pallets. | • Pallets were commanded to move with the clamps opened. |

| DISPLAYED MESSAGE | DESCRIPTION | POSSIBLE CAUSE(S) |
|---|---|---|
| Material handling system fault. | [Failure] Material handling system has faulted. Laser program execution cannot continue. | • Material handling system has a drive fault. See Material Handling System manual. |
| Material handling system not ready. | [Normal] Material handling system safety interlocks are not present. | • Material handling system is turned off. |
| Motor X1 thermostat. | [Failure] Motor thermostat contact has opened, indicating that the motor has overheated. | • Obstruction in coolant lines to motor.<br>If thermostat contact is not open:<br>• Bad wire connection from motor thermostat to I/O PC Board.<br>• I/O Board failure has occurred (see alarm description).<br>• Power failure. |
| Motor X2 thermostat. | [Failure] See “Motor X1 thermostat”. | See “Motor X1 thermostat”. |
| No assist gas pressure detected. | [Normal] Analog feedback signal registered 0 volts when gas was commanded. | • Out of gas / supply turned off.<br>• Bad wire connection from Analog Output PC board (or Analog Input PC board) to Assist Gas valve. |
| Noncontact head has not been calibrated. | [Normal] The height sensor must be calibrated upon startup of CNC, as well as any time the sensor is disconnected. | If sensor was already calibrated,<br>• HSS electronics alarm will force a recalibration. |
| Noncontact Head Installed prox. switch failed. | [Failure] The noncontact head proximity switch opened after a program started with the switch closed. The control displays this System Alarm message after the program stops. Continued operation with the switch open will change the control to Contact Head mode. | • Proximity sensor mounted too far from flange.<br>• Faulty switch.<br>• Faulty cable. |
| Noncontact feedback – check tip wire connection. | [Failure] When the Z-axis was at the top position, the control detected a low voltage level for the height sensor. | • Broken or loose tip wire.<br>• Bad connection in cable.<br>• Bad Z-axis interface board.<br>• Height sensor not calibrated. (See Calibrating Nozzle Standoff in Sec. 7) |
| Noncontact Z-tip touch. | [Normal] The noncontact head electronics indicated nozzle tip contact with material for longer than the Tip Touch Warning Delay specified on the Z-axis page of the Maint. \| Configuration window. | • Pierce material build-up greater than pierce standoff.<br>• Coax cable is not plugged into noncontact head.<br>• Nozzle tip dirty / damaged.<br>• Faulty noncontact transformer. |
| Nozzle breakaway detected. | [Normal] One or more of three proximity switches have indicated head separation from the magnetic flange. High Voltage is disabled. | If head did not breakaway or collide with an object:<br>• Bad wire connection between breakaway proximity switches and EN_GRGT.<br>• 1 or more proximity switches need adjustment to sense head flange.<br>• Power failure. |
| Pallet loader E-STOP - pallet motion is disabled. | [Normal] An external “EMERGENCY STOP” button was pressed on the pallet loader control. | • Bad wire connection between button and CL-707 control. |
| Pallet motion disabled during automatic pallet loading. | [Normal] An attempt was made to jog the pallets while the pallet loading mechanism was in the pallet area. (Automatic Pallet Loader option must be enabled.) | |
| Pallet motion timed out. | [Failure] Upper or lower pallet slowdown limit switch has been actuated for 6 seconds, but IN or OUT limit switch has not tripped. | • Faulty pallet limit switch.<br>• Pallet cams out of adjustment.<br>• Pallet motor controller circuit breaker tripped. |
| Pallet(s) not in a valid position. | [Normal] One pallet must be IN and one pallet must be OUT to run a program. | • Pallets not jogged to end of travel.<br>• Faulty pallet limit switch.<br>• Pallet cams out of adjustment. |
| Pallets are disabled for Lower Pallet Special Function mode (M43). | [Normal] Reminder that Lower Pallet Special function must be disabled before a pallet move is permitted. | • M43 is active and operator attempted to move a pallet. |
| PLC timer overrun. | [Failure] Internal software error. Contact CINCINNATI. | |
| PMAC checksum error. | [Failure] Motion controller firmware error. | |

| DISPLAYED MESSAGE | DESCRIPTION | POSSIBLE CAUSE(S) |
|---|---|---|
| PMAC Controller response time-out. | [Failure] The motion controller is not responding. | • Loose ribbon cable connection.<br>• PMAC PC board not fully seated.<br>• PMAC PC board failed. |
| PMAC Driver (WinRT) not functioning. Exiting application. | [Failure] Windows startup error. | |
| Process interrupted by CYCLE STOP or RESET button. | [Normal] A “CYCLE STOP”, “FEEDHOLD”, or “RESET” button was pressed during one of the following:<br>• Homing.<br>• Part program execution.<br>• Beam “flash” during Lens Centering or Mirror Alignment. | If no button was pressed:<br>• Ball Transfer Remote Station not connected when Ball Transfer option was enabled.<br>• Bad wiring connection to a “CYCLE STOP” or “FEEDHOLD” button located on the console or hand-held pendant. |
| Process monitor detected a loss of cut. | [Normal] The Process monitor has detected excessive light / plasma during the cut process, exceeding the repetition rate specified on the Configuration page. | • Wrong material library was used.<br>• Incorrect setup for chosen material.<br>• Resonator maintenance overdue. |
| Process monitor detected a pierce blowout. | [Normal] The Process monitor sensors detected excessive light/plasma during the pierce operation. (System will retry pierce once, then stop with an alarm). | • Wrong material library was used.<br>• Incorrect setup for chosen material.<br>• Resonator maintenance overdue. |
| Process monitor system failure. | [Failure] The Process monitor sensors are not responding. | • Broken wire connection.<br>• Module has failed. |
| Protective barrier door open | [Normal] The gated fault area behind the machine is not secure (gate is open). This condition is treated the same as a safety gate / safety zone alarm by the control software. | |
| Remote station must be DISABLED to move the axes or pallets via console. | [Normal] An attempt was made to run a program or to perform a homing or jog move while the Remote Station was ENABLED. | • The Remote Station keyswitch was left in the “ENABLED” position after Lens Centering was completed. |
| Remote station must be ENABLED to align the beam. | [Normal] Turn keyswitch on remote pendant to ENABLE. | |
| Retract head before axis move. | [Normal] X/Y motion is not permitted if nozzle is within 1.5” (38 mm) of pallet height and either the RAPID TRAVERSE light is on or the head is not tracking material. This alarm also occurs if head is not up when user selects Program Test mode without Z-axis moves. | |
| Retract head to move pallets. | [Normal] Z-axis moved below home switch while X and Y were homing ***or*** pallet motion was attempted with Z down. | |
| Safety gate open. | [Normal] The cut area enclosure gate is open. | If gate is not visibly open:<br>• Must press RESET to reset safety relay.<br>• Gate switch is not made.<br>• Both doors must be closed to clear alarm. |
| Safety zone has been entered | [Normal] An external proximity safety device (safety mat or area scanner) is indicating that an operator safety zone has been entered. | If safety device is not visibly tripped:<br>• Bad wire connection from device controller to I/O PC board.<br>• Faulty CR1 relay or timer relay.<br>• Faulty safety device / controller.<br>• Power failure |
| Shutter must be UNLOCKED to align the beam or cut. | [Normal] Shutter switch on console set to LOCK. | |
| Tactile sensor fault (electrical connection or follower ring assembly). | [Failure] See Tactile Foot Manual supplement EM-483. | |
| The PMAC Controller was not found. Exiting application. | [Failure] The motion controller is not responding. | • Loose ribbon cable connection.<br>• PMAC PC board not fully seated.<br>• PMAC PC board failed. |
| Tip standoff calibration error. | [Normal] The Automatic Drift Compensation logic has detected more than a 0.015” (0.38 mm) error in the noncontact head calibration. | • Noncontact head not calibrated recently.<br>• Debris on tip.<br>• Material is not metallic or has plastic coating.<br>• Component failure in noncontact head (sliding barrel, cable, etc.). |

| DISPLAYED MESSAGE | DESCRIPTION | POSSIBLE CAUSE(S) |
|---|---|---|
| Tip standoff calibration not completed. | [Normal] The calibration move was interrupted before calibration was completed. | |
| Unable to read PMAC data from the registry. Exiting application. | [Failure] Registry not configured properly. | |
| X1 amplifier fault. | [Failure] The X1 amplifier has sensed a malfunction and has disabled itself. (Drives and High Voltage are disabled.) | • Motor wiring has been shorted.<br>• Amplifier has overheated due to continuous movement in Rapid mode.<br>• Bad wiring connection between amp and console (amp fault wire or 24C).<br>• Other – Set amp selector switch to HOLD FAULT and monitor front of amplifier for failure codes. |
| X1 axis $I^2t$ fault. | [Failure] The average commanded current for this motor has exceeded the recommended limit. | • I157 / I158 / I169 parameters too low.<br>• Motor gap too large.<br>• Program duty cycle too aggressive – decrease jerk setting. |
| X1 fatal following error. | [Failure] The motor did not go where it was commanded. The actual position of the motor may not differ from the commanded position by more than 0.125 inches (3.2 mm). Drives and High Voltage are disabled. | • Incorrect Following Error Limit parameter on the Configuration screen (default: 0.125 in. or 3.2 mm).<br>• X or Y “Jog/Rapid Acc/Dec Rate” set too high on Configuration Screen.<br>• 3-phase input voltage to machine too low.<br>• Faulty encoder.<br>• Faulty wiring on motor or encoder.<br>• Improperly installed magnet track (magnets out of order or improper gap).<br>• Debris on magnet track.<br>• An excessive feedrate was commanded for a long contour move. |
| X1 negative overtravel limit switch tripped. | [Normal] The X1 negative overtravel limit switch was actuated. | If switch not actuated:<br>• Bad wire connection between switch and console.<br>• Defective 60-pin ribbon cable.<br>• Defective opto-isolator on motion control PC board.<br>• Power failure. |
| X1 negative soft overtravel. | [Normal] The position of the motor exceeded the Overtravel Limit setting on the Configuration screen ***or*** a hard overtravel limit switch was actuated. | If occurring during Jog mode:<br>• Overtravel limits not correctly set on Configuration screen.<br>• Clamp Zone parameters not set correctly on Configuration screen.<br>If occurring during Auto mode:<br>• Part program commanding X/Y too close to end of travel. ***Or***<br>• Bad wire connection to overtravel limit switch. |
| X1 positive overtravel limit switch tripped. | [Normal] The X1 positive overtravel limit switch was actuated. | • See “X1 negative overtravel limit switch tripped”. |
| X1 positive soft overtravel. | [Normal] The position of the motor exceeded the Overtravel Limit setting on the Configuration screen ***or*** a hard overtravel limit switch was actuated. | If occurring during Jog mode:<br>• Overtravel limits not correctly set on Configuration screen.<br>• Clamp Zone parameters not set correctly on Configuration screen.<br>If occurring during Auto mode:<br>• Part program commanding X/Y too close to end of travel.<br>Or<br>• Bad wire connection to overtravel limit switch. |
| X2 amplifier fault. | [Failure] See X1 amplifier fault. | See X1 amplifier fault. |
| X2 axis $I^2t$ fault. | [Failure] The average commanded current for this motor exceeded the recommended limit. | • I257 / I258 / I269 parameters too low.<br>• Motor gap too large.<br>• Program duty cycle too aggressive – decrease jerk setting. |

| DISPLAYED MESSAGE | DESCRIPTION | POSSIBLE CAUSE(S) |
|---|---|---|
| X2 fatal following error. | [Failure] See X1 fatal following error. | See X1 fatal following error. |
| X2 negative overtravel limit switch tripped. | [Normal] The X2 negative overtravel limit switch was actuated. | See “X1 negative overtravel limit switch tripped”. |
| X2 negative soft overtravel. | [Normal] See X1 negative soft overtravel. | See X1 negative soft overtravel. |
| X2 positive overtravel limit switch tripped. | [Normal] The X2 positive overtravel limit switch was actuated. | See “X1 negative overtravel limit switch tripped”. |
| X2 positive soft overtravel. | [Normal] See X1 positive soft over-travel. | See X1 positive soft overtravel. |
| X-motor parameters do not match. | [Failure] X1 and X2 speed and acceleration parameters must be identical. (Necessary to maintain gantry alignment when moving.) | • Error in <SerialNo.>i.dat parameter file. |
| Y-amplifier fault. | [Failure] See X1 amplifier fault. | See X1 amplifier fault. |
| Y axis exceeds Lower Pallet Special Function mode (M43) limit. | [Normal] The Y-axis has reached the over-travel limit specified in the Configuration for Lower Pallet Special Function. This limit is considerably smaller than the travel range, to discourage operators from climbing into the machine to load and unload parts. Contact CINCINNATI INCORPORATED if additional travel is required in this axis. | |
| Y-axis $I^2t$ fault. | [Failure] The average commanded current for this motor exceeded the recommended limit. | • I357 / I358 / I369 parameters too low.<br>• Motor gap too large.<br>• Program duty cycle too aggressive – decrease jerk setting. |
| Y-fatal following error. | [Failure] See X1 fatal following error. | See X1 fatal following error. |
| Y-motor thermostat. | [Failure] Motor thermostat contact has opened, indicating that the motor has overheated. | • Obstruction in coolant lines to motor.<br>If thermostat contact is not open:<br>• Bad wire connection from motor thermostat to I/O PC Board.<br>• I/O Board failure (See alarm description).<br>• Power failure. |
| Y negative overtravel limit switch tripped. | [Normal] The Y negative overtravel limit switch was actuated. | See “X1 negative overtravel limit switch tripped”. |
| Y negative soft overtravel. | [Normal] See X1 negative soft overtravel. | See X1 negative soft overtravel. |
| Y positive overtravel limit switch tripped. | [Normal] The Y positive overtravel limit switch was actuated. | See “X1 negative overtravel limit switch tripped”. |
| Y positive soft overtravel. | [Normal] See X1 positive soft overtravel. | See X1 positive soft overtravel. |
| Z amplifier fault. | [Failure] See X1 amplifier fault. | See X1 amplifier fault. |
| Z-axis ABOVE surface reference. | [Normal] Noncontact head raised more than 0.50 inches (12.7 mm) from starting height while tracking material. | |
| Z-axis amp not enabled – press RESET. | [Normal] The “RESET” button must be pressed to reactivate the Z amplifier. This message will always appear following loss of DRIVES power, or certain Z-axis faults. (DRIVES power must be re-established before this message will appear.) | • The DRIVES selector switch was turned to “LOCK/OFF”.<br>• An Auto Shutdown occurred, which automatically turns off DRIVES power.<br>• An X, Y or chiller fault occurred, which automatically turns off DRIVES power.<br>• One of the following alarms occurred: Z-fatal following error, Z-amp fault, Z-soft overtravel, or noncontact head hard overtravel.<br>If Z-axis will not home:<br>• Both the Z-home and Z-down proximity switches are tripped, indicating bad switch operation or wiring. |
| Z-axis BELOW surface reference. | [Normal] Noncontact head lowered more than 0.50 inches (12.7 mm) over a 6.0 inch (152 mm) move while tracking material. | • Material warp is excessive.<br>• Head tracked off material. |

| DISPLAYED MESSAGE | DESCRIPTION | POSSIBLE CAUSE(S) |
|---|---|---|
| Z-axis fatal following error. | [Failure] The motor did not go where it was commanded to go. The **actual** position of the motor may not differ from the **commanded** position by more than 0.125 inches (3.2 mm). | • Incorrect Following Error Limit parameter on the Configuration screen (default: 0.125 in. or 3.2 mm).<br>• “Noncontact Max Accel” machine configuration parameter set too high.<br>• Faulty encoder or brake.<br>• Faulty wiring on motor or encoder.<br>• Z-axis “Max Speed for all Moves” too high (default: 1700 IPM or 43180 mm/min.) |
| Z-axis motion timed out. | [Normal] Z-axis failed to reach final cutting position within 6 seconds of being commanded. | • Material vibrating too much for head to track.<br>• Loose coupling on Z-axis motor. |
| Z-axis negative soft overtravel. | [Normal] The Z-axis Machine Position exceeded the Negative Overtravel limit setting on the Configuration screen.<br>Or<br>The Z-axis moved 0.5” (12.7 mm) below the Top of Pallet setting on the Configuration screen. | • Incorrect Negative Overtravel Limit parameter on Configuration screen (default: -0.5 inches or -12.7 mm).<br>• Top of Pallet setting incorrect or pallet supports need replacement.<br>• Z-axis has “bottomed out” (only possible on lower pallet). |
| Z-axis noncontact hard overtravel. | [Normal] The Y-plate separated from the Z-block during noncontact tracking. | • Counterbalance springs not locked in the DOWN position.<br>• Z-down proximity switch out of adjustment.<br>• Bad wire connection on Z-down proximity switch. |
| Z-axis not homed. | [Normal] The Z-axis has not successfully performed a homing move since the last time the control application was started. | If the Z-axis will not home:<br>• Z-home proximity switch malfunction or bad connection.<br>• Z-down proximity switch malfunction or bad connection. |
| Z-axis positive soft overtravel. | [Normal] The Z-axis Machine Position has exceeded the Overtravel limit setting on the Configuration screen. | • Positive Overtravel Limit parameter not set correctly on Configuration screen (default: 10.0 inches or 254 mm). |
| 110 volt AC power failure. | [Failure] The 110 VAC monitoring relay has detected a loss of power. | |

# PROGRAM ERRORS

| DISPLAYED MESSAGE | DESCRIPTION | POSSIBLE CAUSE(S) |
|---|---|---|
| A pierce dwell time of 0 is specified. Please set 1 in the D argument (G89). | **G103** is not a valid G code if the last **G89** command specified no pierce dwell. See **LASER NC PROGRAMMING** topic. | |
| Brace open/close error. | See **LASER NC PROGRAMMING** topic | |
| Cannot trace through pallet moves. | | |
| Cannot use G31 with cutter compensation on. | See **LASER NC PROGRAMMING** topic | • **G41** or **G42** was active when a program commanded **G31**. |
| Cannot use G84 T4 if Process Parameters pierce mode is "No Pierce" | | |
| Command too long to send to PMAC. | Internal error—call CINCINNATI. | |
| Comment syntax error. | See **LASER NC PROGRAMMING** topic | |
| Corner radius too large. | See **LASER NC PROGRAMMING** topic | |
| Current program is active. Cannot load new program. | | |
| Divide by zero error. | A zero in the denominator is not permitted. | |
| Duplicate line number. | | |
| Duplicate subroutine names in file. | | • More than one subroutine in the current file uses the same name. |
| End of history buffer or beginning of program reached. Reverse tracing is disabled. | Tracing backwards beyond start of program is not possible. In addition, a fixed number of executed program blocks are held in memory while a program is running. This memory area, known as the “history buffer”, limits how far a program can be traced in reverse. | • Large programs contain more blocks than the history buffer can hold. Attempts to trace backwards beyond the last block in the history buffer will cause this alarm. |
| End of program reached. Forward tracing is disabled. | Tracing operation has reached the last line of the program. Press “Trace REVERSE” to back up to desired location for resumption of cut. | |
| Error occurred reading Process Library file. | | • Process Parameter file specified does not exist or is corrupt. |
| G53 ignored in incremental mode. | | |
| G84 or G85 call before a G89. | Cannot start cut until program specifies cutting Process Parameters (power, standoff, etc.) | |
| G89 Call Error. | See **LASER NC PROGRAMMING** topic | |
| G84 T2 or G84 T5 not allowed - Rapid Pierce option is required. | G84 ‘T2’ or ‘T5’ argument is not valid on machines that do not have the optional **Rapid Pierce** function. | |
| G102 call before a G89. | See **LASER NC PROGRAMMING** topic | |
| G103 call before a G89. | See **LASER NC PROGRAMMING** topic | |
| If, Then syntax error. | See **LASER NC PROGRAMMING** topic | |
| Illegal arc radius specified. | | |
| Illegal corner rounding command. | | |
| Illegal exit and entry in the WHILE loop. | See **LASER NC PROGRAMMING** topic | |
| Illegal GOTO statement. | See **LASER NC PROGRAMMING** topic | |
| Incorrect syntax for laser applications. | | NC program contains an invalid statement, or a command argument is missing |
| Invalid data range. | | A G code or M code argument is out of range |
| Invalid laser mode. | | G89 ‘M’ or ‘H’ argument not valid |
| Invalid G code. | See **LASER NC PROGRAMMING** topic | |
| Invalid M code. | See **LASER NC PROGRAMMING** topic | |

| DISPLAYED MESSAGE | DESCRIPTION | POSSIBLE CAUSE(S) |
|---|---|---|
| Line numbering syntax error. | See **LASER NC PROGRAMMING** topic | |
| Look ahead error -- too many lines between motion. | | |
| Main program not found. | See **LASER NC PROGRAMMING** topic | |
| Math statement error. | See **LASER NC PROGRAMMING** topic | |
| Near field pressure exceeds Far field pressure. | | • Dynamic Assist Gas Near field pressure (G102 'A') greater than Far field pressure (G102 'B'). |
| No feedrate specified in program. | See **LASER NC PROGRAMMING** topic | |
| No program name in subroutine call. | See **LASER NC PROGRAMMING** topic | |
| No R, I, or J specified for circle. | See **LASER NC PROGRAMMING** topic | |
| Pallets are disabled for Lower Pallet Special Function mode (M43). | A pallet move was attempted while **M43** was active. Clear parts from pallet and command **M44** before attempting a pallet move. | |
| Parser out of memory. | Internal error—call CINCINNATI INCORPORATED Service Department | |
| Process Parameter library not found. | Process Library file not found | • File name misspelled in program.<br>• File not located in MATERIAL subdirectory. (If file is not in the MATERIAL subdirectory, the G89 call must specify the file/directory path.) |
| Process Parameter out of range error - assist gas port | An invalid Process Parameter setting for assist gas type or port number was encountered by the NC Program parser | • **G89 A** or **B** argument not valid. Valid argument values are 11 (Port #1) or 12 (Port #2)<br>• Process Parameter file specifies an assist gas that is not currently available |
| Process Parameter out of range error - beam size | An invalid beam size Process Parameter was encountered by the NC Program parser | • Out of range Process Parameter file setting |
| Process Parameter out of range error - feedrate | An invalid feedrate Process Parameter was encountered by the NC Program parser | • Out of range Process Parameter file setting or **G89 U** argument |
| Process Parameter out of range error - focus | An invalid focus Process Parameter value was encountered by the NC Program parser | • Out of range Process Parameter file setting, or **G102 Q**, **R**, **U**, or **V** argument |
| Process Parameter out of range error - gas pressure | An invalid gas pressure Process Parameter was encountered by the NC Program parser. Assist gas pressure settings must not exceed Assist Gas Valve Max Rating configuration value | • Out of range Process Parameter file setting or **G89 E** or **I** argument |
| Process Parameter out of range error - kerf compensation | An invalid kerf compensation Process Parameter was encountered by the NC Program parser. | • Out of range Process Parameter file setting or **G89 K** argument |
| Process Parameter out of range error - minimum DPC power | An invalid Minimum DPC Feedrate Process Parameter was encountered by the NC Program parser | • Out of range Process Parameter file setting or **G89 V** argument |
| Process Parameter out of range error - optional assist gas pressure | An invalid optional gas pressure parameter was encountered by the NC Program parser. Optional assist gas pressure setting must not exceed Assist Gas Valve Max Rating configuration value | • Out of range Process Parameter file setting or **G102 I** argument |
| Process Parameter out of range error - pierce dwell time | An invalid pierce dwell time Process Parameter was encountered by the NC Program parser | • Out of range Process Parameter file setting or **G89 D** argument |
| Process Parameter out of range error - Pierce-thru time | An invalid pierce-thru check time Process Parameter was encountered by the NC Program parser | • Out of range Process Parameter file setting |
| Process Parameter out of range error - pre-cut dwell | An invalid pre-cut dwell Process Parameter was encountered by the NC Program parser | • Out of range Process Parameter file setting or **G102 D** argument |
| Process Parameter out of range error - power burst time | An invalid power burst time Process Parameter was encountered by the NC Program parser. | • Out of range Process Parameter file setting or **G102 T** argument |
| Process Parameter out of range error - power ramp tip cooling time | An invalid ramp cooling time Process Parameter was encountered by the NC Program parser. | • Out of range Process Parameter file setting or **G103 F** argument |

| DISPLAYED MESSAGE | DESCRIPTION | POSSIBLE CAUSE(S) |
|---|---|---|
| Process Parameter out of range error - power setting | An invalid ramp step power Process Parameter was encountered by the NC Program parser | • Out of range Process Parameter file setting or **G103 R** through **W** argument. |
| Process Parameter out of range error - pulse duty cycle | An invalid pulse duty cycle Process Parameter was encountered by the NC Program parser. | • Out of range Process Parameter file setting or **G89 R** or **S** argument. |
| Process Parameter out of range error - pulse frequency | An invalid pulse frequency Process Parameter was encountered by the NC Program parser. | • Out of range Process Parameter file setting or **G89 R** or **S** argument. |
| Process Parameter out of range error - ramping time increment | An invalid ramp step time Process Parameter was encountered by the NC Program parser. | • Out of range Process Parameter file setting or **G103 A** through **E** argument. |
| Process Parameter out of range error: resonator wattage | An invalid laser power Process Parameter was encountered by the NC Program parser | • Out of range Process Parameter file setting, or **G89 T** or **Q** argument exceeds nominal power rating of resonator. |
| Program feedrate exceeds machine limits. | The program attempted to command a feedrate greater than the Rapid (G0) Speed. | • The Rapid Traverse Speed listed in Section 4 is not specified on the X Axis and Y Axis pages of the **Maint. \| Configuration** window.<br>• Program feedrates are not commanded in the proper units (IPM or mm/min). |
| Program is missing DPRNT statement(s). | The CNC Serial Output option is enabled AND the program does not command the required number of DPRNT statements. | • Incorrect DPRNT setting on Configuration screen, under Serial Port. |
| Program terminated due to machine setup / Process Parameter conflict | The NC Program parser stopped loading or executing the part program due to a machine set-up conflict. | See **MACHINE SETUP CHECKING** topic |
| Sequence number not found. | See **LASER NC PROGRAMMING** topic | |
| Subprogram call without a return statement. | See **LASER NC PROGRAMMING** topic | |
| Subroutine not found. | Subroutine file not found or misspelled. | • Subroutine name misspelled in program.<br>• File not located in “PROGRAM” subdirectory. (If file is not in “PROGRAM” subdirectory, the program must specify the file/directory path.) |
| Too many nested IF statements. | See **LASER NC PROGRAMMING** topic | |
| Too many ramping power steps specified. | G103 ‘Q’ argument not valid. (For up to 20 ramping steps, use G89 to call a material library file instead of using G103.) | • G103 ‘Q’ commanded more than 3 ramping steps. |
| Too many nested subroutines. | See **LASER NC PROGRAMMING** topic | |
| Too many nested WHILE loops. | See **LASER NC PROGRAMMING** topic | |
| Unable to open Process Library file. | | • Process Parameter file specified does not exist or is corrupt |
| Variable assignment error. | See **LASER NC PROGRAMMING** topic | |
| WHILE loop syntax error. | See **LASER NC PROGRAMMING** topic | |
| Workpiece Edge Detection option must be enabled for G31. | G31 cannot be used on machines that do not have the optional **Workpiece Edge Detection** function. | • Option not enabled on Configuration “Options” screen. |
| X axis travel exceeds machine limits. | The selected program will not fit on the table if started at the present location. | • Incorrect Soft Overtravel limit machine configuration setting |
| Y axis travel exceeds Lower Pallet Special Function mode (M43) limit. | | • Incorrect LPSF Soft Overtravel limit machine configuration setting |
| Y axis travel exceeds machine limits. | The selected program will not fit on the table if started at the present location. | • Incorrect Soft Overtravel limit machine configuration setting |
| Z-axis noncontact data error. | An invalid standoff Process Parameter was encountered by the NC Program parser | • Out of range Process Parameter file setting or G102 ‘S’ or ‘Z’ argument. |

# LASER ALARMS

| DISPLAYED MESSAGE | DESCRIPTION | POSSIBLE CAUSE(S) |
|---|---|---|
| HSSB error (*number*).<br>(See examples that follow.) | An error occurred in the communications link between the CINCINNATI CNC control and the resonator control. The HSSB (high speed serial bus) interface board connects the CINCINNATI control to the fiber-optic communications cable. | • Serial communication link is broken or disconnected.<br>• FANUC resonator control is not running.<br>• HSSB interface board malfunction.<br>• FANUC resonator control is not configured properly, or is malfunctioning. |
| HSSB error (–7). | Software version mismatch. See resonator manual. | |
| HSSB error (–3). | FANUC drivers are not installed. See resonator manual. | |
| HSSB error (12). | Incorrect CNC mode. See resonator manual. | |
| 4061 A/D CONVERTER-1 | Abnormal condition in A/D converter 1 (mount B8 on interface board). See resonator manual. | • Failed converter or board.<br>• Fanuc CNC parameters cleared. |
| 4062 A/D CONVERTER-2 | Abnormal condition in A/D converter 2 (mount D7 on interface board). See resonator manual. | • Failed converter or board.<br>• Fanuc CNC parameters cleared. |
| 4063 RF POWER SUPPLY | The power supply reported an error or performed a protective operation. See resonator manual. | See resonator manual. |
| 4065 SHUTTER ACTION | The shutter did not operate normally for the open/close command. | See resonator manual. |
| 4066 DISCHARGING | Abnormal discharge caused high voltage. | See resonator manual. |
| 4067 LASER CABINET OH | Temperature at electrode cooling fan inlet exceeds 60 deg. C (140 F). | See resonator manual. |
| 4068 BEAM REFLECTION | Energy reflected back to the resonator exceeds alarm limit (parameter). | See resonator manual. |
| 4069 LASER IF PCB | Abnormal interface board power voltage (+5, +/- 15, +24V) | See resonator manual. |
| 4070 CHILLER NOT READY | The Laser System did not send the Chiller Ready signal to the resonator control. | See resonator manual. |
| 4072 CHILL FLOW | Cooling water flow is below alarm limit. | See resonator manual. |
| 4073 LASER GAS PRES. | Laser gas supply pressure is below alarm limit. | See resonator manual. |
| 4075 CHILL TEMP. | A sensor detected condensation, indicating low cooling water temperature. | See resonator manual. |
| 4076 LASER POWER DOWN | Laser output power is less than commanded power by more than alarm limit (parameter 15271). | See resonator manual. |
| 4077 ABSORBER TEMP. | Beam absorber temperature is too high. | See resonator manual. |
| 4078 LASER TUBE PRES. | Laser gas pressure control error is more than the alarm limit. | See resonator manual. |
| 4079 PUSH RESET KEY | This message reminds the operator to press RESET after releasing the “EMERGENCY STOP” button. | |
| 4080 LASER TUBE EXHAUST | Too much time was required for the exhaust pump to lower the pressure in the resonator. | See resonator manual. |
| 4081 GAS PRES. CONTROL | After back-filling the discharge tubes with laser gas, the pressure was not controlled within the required tolerance. | See resonator manual. |
| 4082 TUBE PRES. SENSOR | No pressure sensor signal. | See resonator manual. |
| 4085 MIRROR CLEANING | Laser output power decreased, requiring an abnormally high calibration coefficient. | • Internal optic contamination or misalignment.<br>• Incorrect laser gas composition, cooling water temperature, or ambient temperature.<br>• See resonator manual. |
| 4087 SHUTTER OH | Shutter mirror temperature sensor indicated an over-heated condition. | See resonator manual. |
| 4088 LASER VOLTAGE DOWN | The discharge tube voltage required at startup (parameter 15270) is less than the previous value by more than a limit (parameter 15272). | See resonator manual. |
| 4090 LASER NOT GENERATE | Laser operation was attempted when the resonator was not in the discharging state. | |

| DISPLAYED MESSAGE | DESCRIPTION | POSSIBLE CAUSE(S) |
|---|---|---|
| 4094 VANE PUMP | Exhaust pump overload (motor contactor thermal switch tripped). | See resonator manual. |
| 4099 GAS PRES. NOT REACH | After starting High Voltage, the discharge tube gas pressure was not controlled within the required tolerance. | See resonator manual. |
| 4100 INVERTER ABNORMAL 1 | Turbo blower inverter issued an alarm. Check inverter LED indicator. | See resonator maintenance manual. |
| 4101 INVERTER FREQUENCY REACHED SIGNAL ABNORMAL | Turbo blower inverter did not reach required frequency after 120 seconds. | See resonator manual. |
| 4105 TURBO BLOWER ABNORMAL TEMPERATURE | Turbo blower motor winding temperature exceeded alarm limit. | See resonator manual. |
| 4106 TURBO BLOWER OIL LOW | Turbo blower sensor indicated low oil level. | See resonator manual. |
| 4107 EXTERNAL FOLDING MIRROR NOT INSTALLED | A sensor indicated that the folding mirror is not installed. | See resonator manual. |
| 4132 PARAMETER IS CHANGED | A parameter was changed. Press RESET to clear the alarm. | |
| Laser serial port configuration failure. | CNC could not initialize resonator interface COM port. | • PC Serial COM port used for CNC/ resonator interface is not configured properly or is malfunctioning. |
| Laser serial port could not be opened. | CNC could not open resonator interface serial COM port. | See “Laser serial port configuration failure”. |

# TROUBLESHOOTING

Effective and safe troubleshooting procedures are acquired through experience and a thorough knowledge of the machine and its operation. The use of maintenance instructions, assembly drawings, and schematics included with the Laser System **Operation, Safety, and Maintenance Manual** will be helpful in resolving problems with the machine.

The following chart was developed to aid in troubleshooting problems with the machine. The chart contains questions frequently asked by our customers. If major repairs are required or if the problem has not been identified by using the chart, contact the Service department at CINCINNATI INCORPORATED for assistance. Only qualified personnel should attempt to troubleshoot electrical systems.

| PROBLEM | POSSIBLE CAUSE | REPAIR |
|---|---|---|
| Control will not turn on. | No power to machine. | • Verify line voltage.<br>• Check condition of fuses / disconnect. |
| | Power supply failure. | • Verify power supply voltage.<br>• Check condition of fuses. |
| | UPS (uninterruptible power supply) turned off. | • If the UPS was turned off manually, it must be turned on before the control will start. |
| Cannot start cycle. | Alarm condition present.<br>Improper control configuration. | • Check alarm status and correct any conditions present.<br>• Verify correct operating mode and program status. |
| Intermittent loss of cut | Beam disabled due to loss of material contact. | • Verify operation of Z-axis switches.<br>• Check surface condition of material. |
| Laser will not turn on. | No power to Laser. | • Verify line voltage.<br>• Check condition of line fuses in machine electrical panel.<br>• Check condition of fuses and circuit breakers in Laser cabinet. |
| | “Mains On” command not sent from control. | • Check connection of interface cable.<br>• Verify operation of switch. |
| HV will not turn on or will not stay on while cutting. | Nozzle interlocks not properly made. | • Reseat cutting head assembly on flange. |
| | High Voltage keyswitch on Operator Control Station side panel is in the LOCK/OFF position | • Set High Voltage keyswitch for proper operation. |
| | Shutter not in closed position or did not achieve proper position during operation. | • Verify proper operation of shutter mechanism and position indicator switches. |
| **Change in cut quality.** | See **Process Parameters and Diagnostic Guide**. | |

# LASER NC PROGRAMMING

## STANDARD G CODES

See **G Code List** for a table of all G codes recognized by the Laser System CNC.

The following four G codes all move the cutting nozzle to commanded Work coordinates:

| | |
|---|---|
| **G00** | Rapid Traverse move |
| **G01** | Linear move |
| **G02** | Clockwise Arc |
| **G03** | Counterclockwise Arc |

These four G codes form a modal group; the last G code commanded in the group is active for all blocks until the program commands another G code in the group. The default code when a program starts is **G00**. The leading zero can be omitted; **G0**, **G1**, **G2**, and **G3** are the same as **G00**, **G01**, **G02**, and **G03**.

Each of these G codes specifies the end of the move with X and Y in the Work coordinate system. X and Y are absolute coordinates when the program commands the block in **G90** mode, and incremental distances when commanded in **G91** mode. The command must specify at least one axis.

The **G00** command moves the axes at the rapid traverse rate of the machine. **G01**, **G02** and **G03** move the axes at the contouring feedrate (optionally specified in the block with **F**). When the block does not command a feedrate, the program uses the last defined contouring feedrate. When the control applies the rapid traverse rate for a **G00** move, it does not change the contouring feedrate used by the **G01**, **G02**, and **G03** blocks.

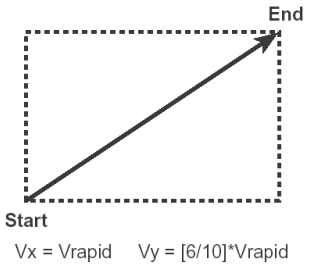
### G00 RAPID TRAVERSE MOVE

**G00 X__ Y__** Rapid Traverse move

The **G00** command moves the cutting nozzle to a work coordinate location (or incremental distance) using the rapid traverse rate. When the command requires both axes to move, the axis moving the longer distance uses the rapid traverse rate of the machine. The other axis moves at a lower velocity proportional to the distance required, so both reach their endpoints at the same time, approximating linear interpolation.

*Example:*

**(G91) G00 X10 Y6**

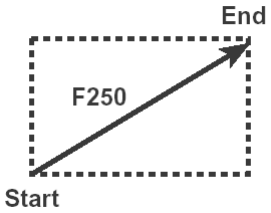


## G01 LINEAR MOVE

**G01 X__ Y__ (F__)** Linear move

This command moves the cutting head to the work coordinates (or incremental distance) defined by X and Y, at a contouring feedrate optionally specified by **F**.

*Example:* **(G91) G01 X6 Y4 F250**



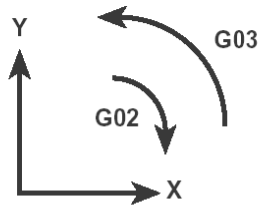

When the command requires both axes to change position, the machine moves each axis at a velocity required to produce a combined feedrate equal to the contouring feedrate. The move follows the linear path between start and end points.

## G02 AND G03 ARC MOVE

**G02 X__ Y__ I__ J__ (F__)** Clockwise Arc move (offsets specified)
**G02 X__ Y__ R__ (F__)** Clockwise Arc move, (radius specified)
**G03 X__ Y__ I__ J__ (F__)** Counterclockwise Arc move, (offsets specified)
**G03 X__ Y__ R__ (F__)** Counterclockwise Arc move, (radius specified)

A program uses **G02** (clockwise arc) or **G03** (counterclockwise arc) to command a circular contouring move ending at the work coordinates (or incremental distances) specified by X and Y. The command defines the shape of the arc either by specifying incremental distances (with **I** and **J**) from the starting position to the center, or by specifying the radius (with **R**). The control software interprets **I** and **J** as distances in the X and Y directions (respectively) from the starting position to the center. When the command specifies radius **R**, the control moves the nozzle along a circular path with that radius.

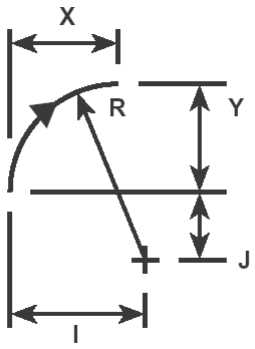
The machine maintains the modal contouring feedrate **(F**) along the circular path.




G02 X_ Y_ I_ J_ (F_)
G02 X_ Y_ R_ (F_)
G03 X_ Y_ I_ J_ (F_)
G03 X_ Y_ R_ (F_)

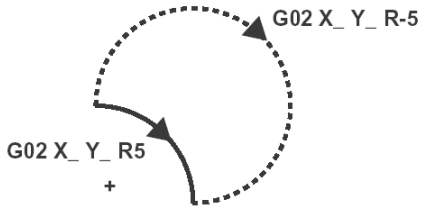
*Example:*

**(G91) G02 X5 Y4 I7 J-3**




When the block uses **R** instead of **I** and **J,** there are two possible arcs for a given direction (CW or CCW) and end coordinates. To specify which arc to contour, the block commands **R** with a positive or negative sign. To specify an arc that is less than 180 degrees, the block commands a positive **R** value. To specify an arc greater than 180 degrees, the **G02** or **G03** block commands **R** with a negative value.

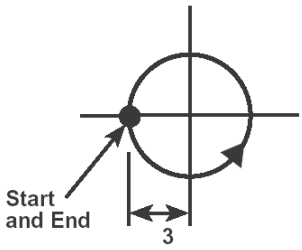
*Example:*




When **G02** or **G03** specifies the same coordinates for the start and end of the arc, the machine contours a complete circle. For complete circles, the block must specify the center with **I** and **J.** Programming software must specify both coordinates accurately. If the ending coordinates for a circular move are not exactly the same as the starting coordinates, the path may be a very small arc instead of a complete circle. To avoid this problem, programs can omit **X** and **Y** from a **G02** or **G03** block to command a complete circle; the control will automatically apply the same starting and ending coordinates.

*Example:*

**(G91) G03 I3 J0**




### Recommended Arc Feedrate

Recommended maximum **G02** or **G03** feedrate depends on machine design, arc radius, and allowable roundness error. Use this equation to calculate the maximum feedrate for each arc:

$$F = K * \sqrt{R * (T - T_0)}$$

**F** = Arc feedrate (IPM or mm/min.)

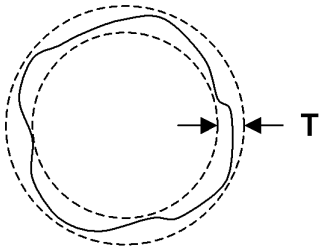
**K** = Constant (1 / min.) See tables.

**R** = Arc radius (inches or mm)

**T** = Roundness tolerance (inches or mm)

**$T_0$** = Minimum radial error (inches or mm)

Roundness tolerance **T** is the radial distance between two concentric circles that enclose the contoured shape. To use this formula, the specified roundness tolerance must be greater than **$T_0$** and not more than **$T_{max}$**.




The maximum acceleration also determines the maximum feedrate for contouring an arc. The following tables include that requirement by specifying a maximum roundness **$T_{max}$** for each value of **K**. If the roundness tolerance does not exceed **Tmax**, then the calculated feedrate will not command the machine to exceed the maximum acceleration.

| CL-800 Arc Feedrate Programming | | | |
|---|---|---|---|
| Machines built July, 1999 and later ("Fast Pack" Drive Design) | | | |
| **Model** | **K** | **$T_0$** | **Tmax** |
| 4x8 | 26,500 | .0002 in. (.005 mm) | .004 in. (.102 mm) |
| 5x10 | 26,500 | .0002 in. (.005 mm) | .003 in. (.076 mm) |
| 6x12 | 26,500 | .0002 in. (.005 mm) | .003 in. (.076 mm) |

To determine the feedrate for contouring an arc, compare the calculated maximum feedrate to a minimum arc feedrate (typically 30 IPM) and select the higher value. Then compare the selected value to the material feedrate, and use the lower value.

## G04 DWELL

**G04 P__** Program Dwell

The **G04** (or **G4**) command causes NC program execution to dwell for the time specified by the **P** argument (in milliseconds). This dwell time does not include the block processing time of the CNC command.

*Example (to dwell for one second):* **G04 P1000**

## G09 EXACT STOP (ONE BLOCK)

**G09** Use Exact Stop for one block

The program commands **G09** (or **G9**) in the same block as a **G00**, **G01**, **G02** or **G03** command. When the block contains **G09**, the control does not proceed to the next block until the axes reach zero feedrate. If the block does not contain **G09**, the control proceeds to the next block when each axis position is within a critical distance of the commanded position.

*Example:* **(G01 X_ Y_ ) G09**

## G20 INCH MODE, G21 METRIC MODE

**G20** Use Inch mode
**G21** Use Metric mode

The **G20** command puts the CNC in the inch units mode. In **G20** mode, the control interprets program coordinates and feedrates in inch system units. (Positions are in inches and feedrates are in inches per minute).

The **G21** command puts the CNC in the metric units mode. In **G21** mode, the control interprets program coordinates and feedrates in metric system units. (Positions are in millimeters and feedrates are in millimeters per minute).

The default mode is **G20** when the **CNC/HMI** application starts. After the control runs a program, the default mode is the same as the last program. To make sure the control interprets a program correctly, the program should begin by commanding **G20** or **G21** to specify units.

**G20** and **G21** do not change the units mode of the **CNC/HMI** application. All user interface windows display values in inch or metric units as selected by **View | Units**.

## G31 POSITION CAPTURE MOVE

**G31 X__ Y__ (F__)** Position Capture move

When a program commands **G31**, the X- and Y-axes move to the specified coordinates in the Work coordinate system. The **G31** command uses the modal contouring feedrate (**F**). While the axes are moving, the control system monitors the *Position Capture* input. If the control system receives the *Position Capture* input, it records the X-axis and Y-axis machine coordinates at that time and stores the values in system variables #5061 and #5062.

If the control detects more than one *Position Capture* input during the move, it only saves the coordinates of the first occurrence. If the control does not receive the *Position Capture* input, it stores the coordinates at the end of the move. The control always completes the move to the coordinates specified in the **G31** block (unless an overtravel alarm stops motion).

Position Capture system variables:

**#5061** = X-axis Machine Coordinate
**#5062** = Y-axis Machine Coordinate

CINCINNATI macro programs use **G31** to find coordinates associated with optional measurement functions (**Workpiece Edge Detection**). The machine control does not accept the **G31** command unless the machine configuration includes one of those options.

## G40, G41, AND G42 KERF COMPENSATION

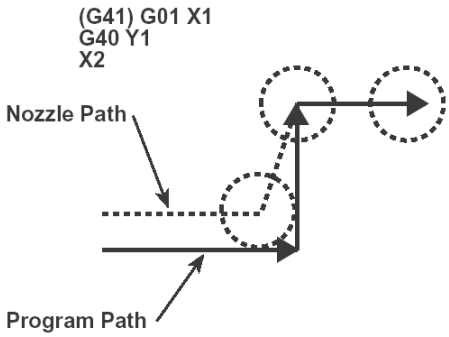
**G40** Cancel kerf compensation
**G41** Apply left side kerf compensation

**G42** Apply right side kerf compensation

**G40** cancels **G41** or **G42**. The cutting nozzle moves from the compensated position to the commanded coordinates during the **G40** move.

*Example:*

**G40**



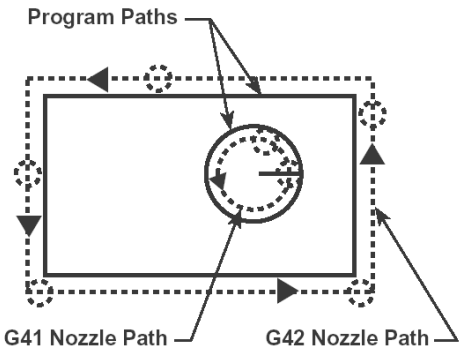

The control automatically cancels kerf compensation at the end of any **G00** or **G53** move if the program commands **G00**, **G53**, **M02,** or **M30** in the next block.

If a program commands **G40** in a block by itself, and then commands a move without **G41** or **G42**, the control cancels compensation during that move.

A program commands kerf compensation with **G41** or **G42**. When a **G01**, **G02** or **G03** block commands **G41** or **G42**, the control begins that move with the nozzle offset to one side of the programmed path. If a block commands **G41** or **G42** without commanding a move, the control ends the previous move with the cutting nozzle offset to one side of the path.

*Example:*

**G41** and **G42**

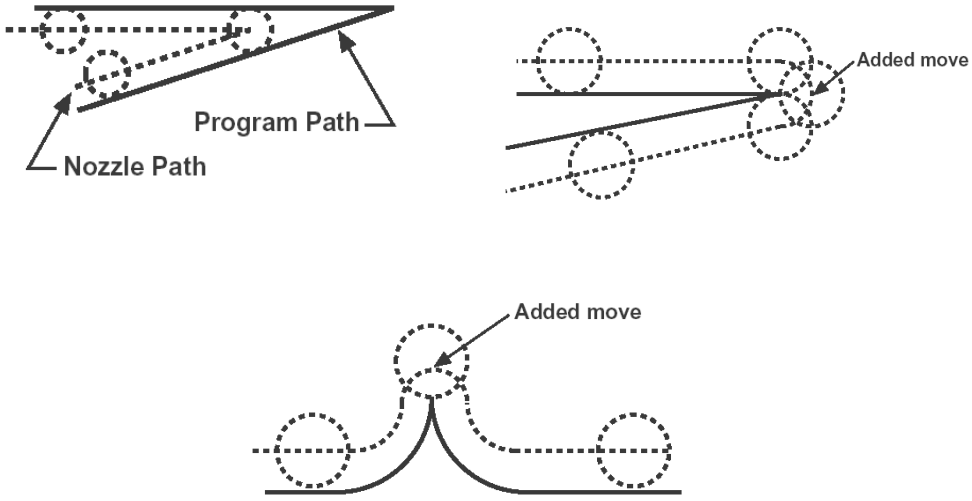



The CNC automatically offsets the cutting nozzle by half the kerf width specified by last **G89** command. (See **CUT PARAMETERS**.)

**G40**, **G41** and **G42** form a modal group; the last G code commanded in the group is active for all blocks until the program commands another code in the group. When each program starts, the default code is **G40**.

The CNC automatically commands the closest possible position for the nozzle to contour the programmed shape with the specified kerf size. If necessary, the control inserts small moves so compensated paths intersect and do not over-cut the shape.

*Examples:*




## G50 AND G51 COORDINATE SYSTEM SCALING

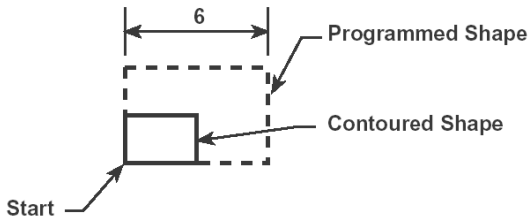
| | |
|---|---|
| **G50** | Cancel Work Coordinate System scaling |
| **G51 X__ Y__ P__** | Scale Work Coordinate System |
| **G51 X__ Y__ I__ J__** | Scale Work Coordinate System |

The control interprets the work coordinate system at a different scale or as a mirror image when the program commands **G51**. The program can restore the normal scale by commanding **G50**. When each program starts, the default mode is **G50**. The *Absolute Position* window and system variables indicate the actual position.

The **G51** block defines the center of scaling with X and Y, and the scale factor with **P**, **I** or **J**. To command 1.0 scale (where the contoured shape is the same as the programmed shape), the **G51** block uses **P1000** (or **I1000** or **J1000**). The **G51** block can use **I** and **J** to command separate scale factors for the X- and Y-axes (respectively). To contour a mirror image of the programmed shape, the block commands **I** or **J** with a negative value. The control does not scale the kerf compensation offset distance when the program commands scaling.

*Example 1:*

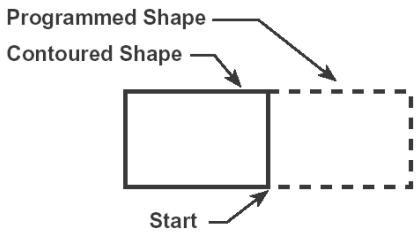
```
G91
G51 X0 Y0 P500
G01 X6
Y4
X-6
Y-4
G50
```




*Example 2:*

```
G91
```

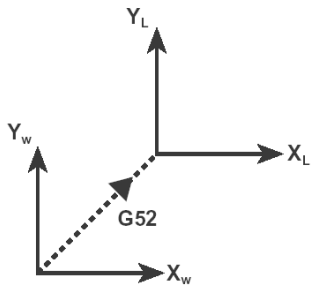
```
G51 X0 Y0 I-1000
G01 X6
Y4
X-6
Y-4
G50
```




## G52 LOCAL WORK COORDINATE SYSTEM

**G52 X__ Y__** Define Local Work Coordinate System

The **G52** command temporarily defines a new work coordinate system while remembering the original. The zero position of the new (or "local") coordinate system is at the coordinates in the original coordinate system specified by X and Y in the **G52** block. After the **G52** block, the program makes contouring moves using the new coordinate system. To restore the original work coordinate system, the program commands **G52 X0 Y0**.




The **G52** block does not move the cutting nozzle. The *Absolute Position* window changes to indicate the nozzle position in the temporary coordinate system.

To demonstrate how a program could use **G52**, consider a program that uses a subprogram to contour the same shape several times, and both the main program and subprogram use **G90** (absolute) mode. The main program would command a work coordinate system with **G92** and the subprogram would command a local coordinate system with **G52**, then cancel it with **G52 X0 Y0**.

## G53 RAPID MOVE TO MACHINE COORDINATES

**G53 X__ Y__** Rapid Move to Machine Coordinates (X Y)

The **G53** command moves the cutting nozzle at the rapid traverse rate to a position specified by **X** and **Y** in the machine coordinate system. **G53** is only active in one block and only in **G90** absolute mode. No motion occurs if the program commands **G53** in **G91** (incremental) mode. The control does not change the machine coordinate system when the program commands kerf compensation, rotation, scaling, or mirror image, or if the program changes the work coordinate system.

## G54…G59 WORK COORDINATE SYSTEM SELECTION

| | |
|---|---|
| **G54** | (Use Work Coordinate System 1) |
| **G55** | (Use Work Coordinate System 2) |
| **G56** | (Use Work Coordinate System 3) |
| **G57** | (Use Work Coordinate System 4) |
| **G58** | (Use Work Coordinate System 5) |
| **G59** | (Use Work Coordinate System 6) |

A program can use **G54** through **G59** to command one of six different pre-defined work coordinate systems. The user can set the distance from Machine X0 Y0 to the Work X0 Y0 position of each coordinate system in the **Work Offsets** window, or the program can assign the distance with system variables #2501 through #2506 (**X**) and #2601 through #2606 (Y).

A work coordinate system defined with **G54** through **G59** does not need **G92** to define its X0 Y0 position. **G54** through **G59** override **G92** by commanding a work coordinate system with its X0 Y0 position preset on the machine.

The **G54** through **G59** block does not move the cutting nozzle. The *Absolute Position* window changes to indicate the nozzle position in the new work coordinate system.

## G61 AND G64 EXACT STOP MODE

| | |
|---|---|
| **G61** | Use Exact Stop mode |
| **G64** | Cancel Exact Stop mode |

**G61** commands the CNC to use Exact Stop mode. In this mode, the axes decelerate to a stop at the end of every **G00**, **G01**, **G02** or **G03** block. The CNC remains in **G61** mode until the program commands **G64** or the program ends.

The **G64** command cancels Exact Stop mode. The default mode when each program starts is **G64**. In **G64** mode, the control proceeds to the next block when each axis position is within a critical distance of the commanded position.

## G65 SUB-PROGRAM CALL

**G65 P__ (A__ B__ C__ D__ etc…)** Sub-Program call (with optional arguments)

The **G65** block specifies the subprogram name after **P**, and may use other arguments to set local variables in the subprogram.

The **G65** block must include a **P** argument, followed by the name of the subprogram. If the subprogram is in the same file as the CNC program, then the subprogram name does not need an extension or path. However, if the subprogram is in a separate file then the **G65** block must include a **P** argument followed by the subprogram file name including its extension (if any) and its path if different from the calling program.

If the **G65** command includes optional arguments, the command must have a space between the last character of the program name and the first argument. This is required because program names can contain both numerals and alphabetic characters. For more on calling subprograms with **G65**, see the Program Structure topic.

## G68 WORK COORDINATE ROTATION

| | |
|---|---|
| **G68 X__ Y__ R__** | Rotate Work Coordinate system |
| **G69** | Cancel Work Coordinate System rotation |

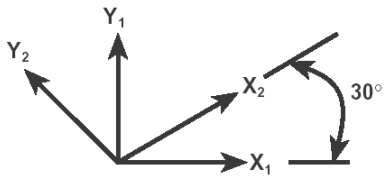
A program can use the **G68** command to rotate the work coordinate system relative to the machine axes. The command specifies the center of rotation with X and Y work coordinates (or incremental distances). The command specifies the amount of rotation with argument **R** in degrees, with counterclockwise positive. In **G90** mode, **R** is the absolute angle of rotation. In **G91** mode, **R** is the incremental rotation angle that the control adds to any previous rotation.

The work coordinate system remains rotated until the program commands **G69** or the program is reset. **G69** cancels all coordinate rotation. To cancel only the last incremental rotation, command **G68** in **G91** mode with the opposite amount for **R**.

The **G68** or **G69** block does not move the cutting nozzle. The *Absolute Position* window and System Variables indicate the nozzle position in the unrotated work coordinate system.

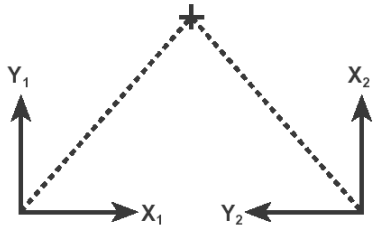
*Example 1:*

**G68 X0 Y0 R30**




*Example 2:*

**G68 X5 Y5 R90**




## G90 AND G91 ABSOLUTE AND INCREMENTAL MODE

| | |
|---|---|
| **G90** | Use Absolute mode |
| **G91** | Use Incremental mode |

In **G90** Absolute mode, the nozzle moves to the coordinate location specified by the X and Y arguments in a **G00**, **G01**, **G02**, **G03,** or **G53** command. **G90** mode is active until the program commands **G91** mode. When each program starts, the default mode is **G90**.

In **G90** mode, X and Y coordinate values are modal. In other words, if a block does not specify X or Y positions, the control uses the last commanded value for X or Y.

In **G91** Incremental mode, the cutting nozzle moves a distance from its starting location specified by X and Y in a **G00**, **G01**, **G02,** or **G03** command. **G91** mode is active until the program commands **G90** or the program ends. The control ignores **G53** commands while operating in **G91** mode.

## G92 WORK COORDINATE SYSTEM SETTING

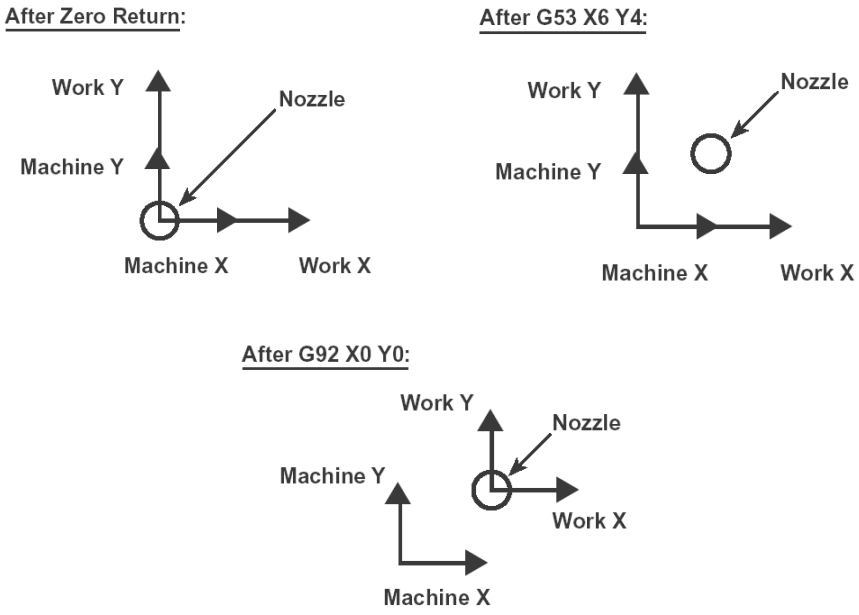
| | |
|---|---|
| **G92 X__ Y__** | Set Work Coordinate System location |

This command sets the work coordinate system location. When the machine completes the Axes Home operation, the control establishes the work coordinate system with X0, Y0 at Machine X0, Y0. The **G92** command can move the work coordinate system to any location. **G92** arguments X and Y define the new work coordinates corresponding to the cutting nozzle position when the **G92** block is executed. The **G92** block does not move the cutting nozzle.

The **G92 X0 Y0** command moves the work coordinate system X0 Y0 location to the current position of the cutting nozzle. Programmers often use this command to begin a subprogram written in **G90** mode.

*Example:*

**G92 X0 Y0**




## CUSTOM G CODES

The CINCINNATI Laser System control has several built-in functions programmed with custom G Codes.

| | |
|---|---|
| **G84** | Pierce and Start Cut |
| **G85** | Start Cut without Pierce |
| **G89** | Set Process Parameters |
| **G102** | Set Additional Parameters |
| **G103** | Ramped Pierce Parameters |
| **G120** | Disable Non-Stop Cutting |
| **G121** | Enable Non-Stop Cutting |
| **G123** | Programmable Blend |
| **G124** | Default Blend |
| **G125** | Auto Blend |

### G84 AND G85 START CUTTING SEQUENCE

| | |
|---|---|
| **G84 T _** | Execute pierce sequence, then start cutting |
| **G85** | Start cutting without a pierce sequence |

**G84** initiates a pierce sequence followed by a start-cutting sequence. During the pierce sequence**,** the cutting head moves down to the pierce standoff position (if not already there), and the beam turns on with the pulse, power, focus and assist gas settings for piercing as specified by the most recently executed **G89** command. When the pierce sequence is complete, the cut sequence begins applying the standoff, power, assist gas and focus settings for cutting and starting a pre-cut dwell, if specified. When the standoff and focus positions are stable and the pre-cut dwell time has expired, **G84** processing is complete and NC program execution resumes with the laser beam on, and shutter open, ready for contouring commands (**G01, G02, G03**).

Use **G85** to begin a start-cutting sequence when the application does not require the pierce sequence of **G84**. **G85** duplicates

all other functions of **G84**, including pre-cut dwell and power burst time (see **G102** description). After a program commands processing parameters with **G89**, any cut sequence can start with **G84** or **G85**. Examples of **G85** applications are: starting a cut inside an opening, off the edge of the sheet, or in a kerf.

### G84 T Argument Options

The optional **T** argument can be used to modify the **G84** command processing as follows:

**G84 T1 - Normal Pierce / Cut**. The **T1** argument is ignored, but does not cause a program error, for older program compatibility. **G84 T1** is identical to **G84.**

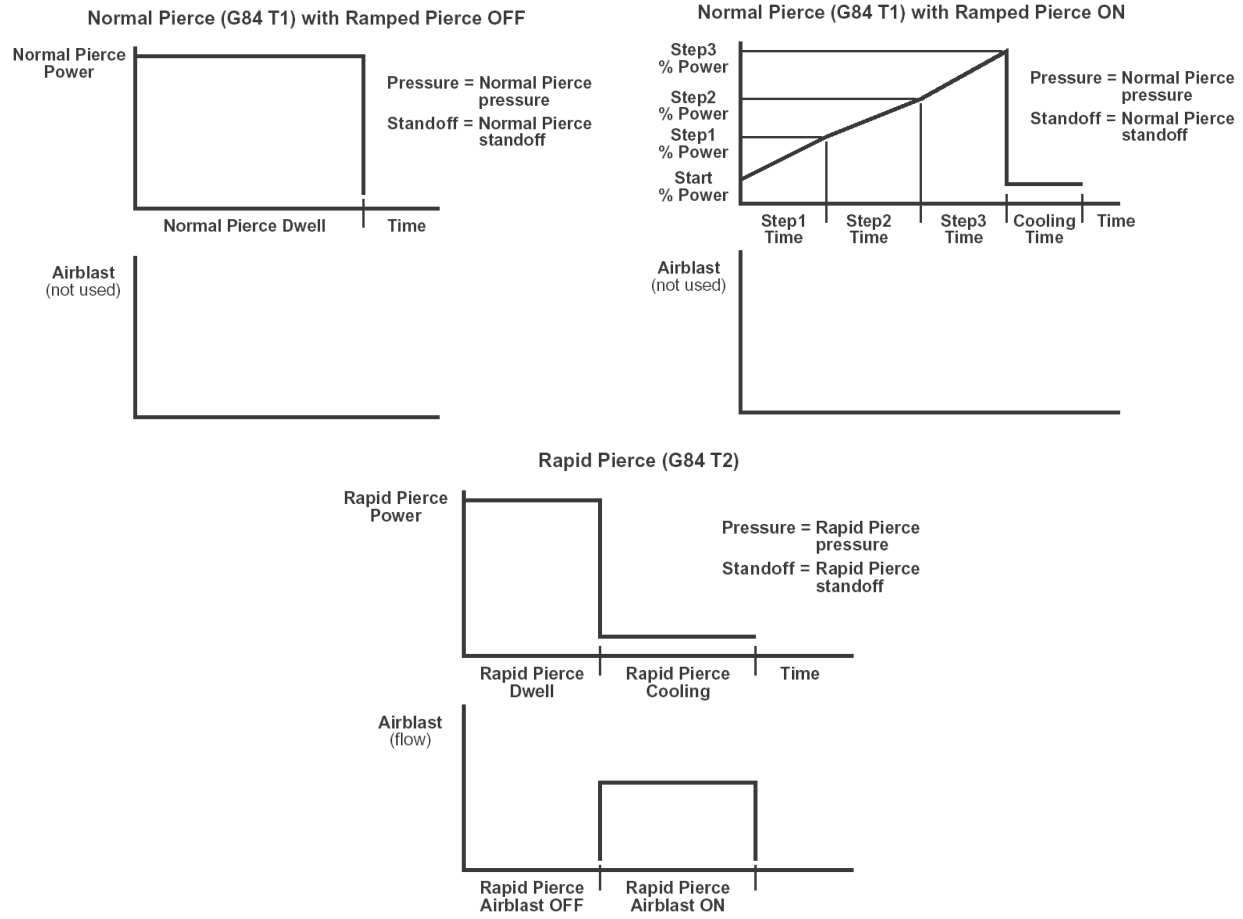
**G84 T2 - Rapid Pierce / Cut**. Use **G84 T2** to command a Rapid Pierce followed by a start-cutting sequence. The Rapid Pierce process uses a blast of compressed air to clear molten material from the pierce area, and has laser power, assist gas pressure, dwell time and standoff parameters that are distinct from the normal pierce settings. See Rapid Pierce Parameters in the Cutting Process Parameters topic for more on Rapid Pierce.

**G84 T3 - Start cutting with no pierce**. This is the same as **G85** (no pierce). The user can also start cutting without piercing by loading Process Parameters (using **G89 P___**) from a file that has Pierce Mode set to "No pierce". (See **PIERCE PARAMETERS** in the **CUTTING PROCESS PARAMETERS** topic.)

**G84 T4 - Pierce only**. When the **T4** argument is used with **G84,** there is no start-cutting sequence performed. In this case, **G84** processing is complete and NC program execution resumes as soon as the pierce sequence is complete. **G84** can be used when it is beneficial to execute all of the pierces in a program before cutting is started.

**G84 T5 - Rapid Pierce only**. This is the same as **G84 T4**, with a Rapid Pierce instead of a normal pierce sequence.

The following figures show the function of **G84 T1** and **T2** parameters:



## G89 SET PROCESS PARAMETERS

| | |
|---|---|
| **G89 P filename** | Set all Process Parameters (file specified) |
| **G89 T__ A__ I__** …etc… | Set specified Process Parameters (using command line arguments) |

Laser NC cutting programs use **G89** commands to specify piercing/cutting Process Parameters. There are two ways to use **G89**. The preferred method is to use the **P** argument to specify the name of a Process Parameter File containing all of the piercing/cutting parameter settings. The second method is to use command line arguments to set specific parameters.

### G89 Using a Process Parameter File

The **G89** command uses the **P** argument to specify a Process Parameter file. Note that all Process Parameter file names must end with the ".lib" extension.

*Example:* **G89 PSS120N2.lib**

The default path for parameter files is: "*C:\CncLsr32\Material*". Other parameter directories can be created in either the "Material" folder or elsewhere on the disk. However, if a Process Parameter file is not in the default parameter folder, the **G89 P** command must include the path of the file with the file name.

*Example:* **G89 P C:\MondayDemo\Material\MS625O2.lib**

CINCINNATI INCORPORATED provides a library of read-only Process Parameter files in the folder "*C:\Cnclsr32\Material\Archive*". As shipped from the factory, the default "Material" directory contains copies of the archive library files, which can be edited by the machine operator.

The names of the files in the Process Parameter library provided by CINCINNATI INCORPORATED use the following conventions:

1. The first part of the file name is an abbreviation for material:

   **AL** = Aluminum
   **MS** = Mild Steel
   **SS** = Stainless Steel

2. After the material abbreviation, a three-digit number represents the material thickness in mils.

   *Example (For 10 gage mild steel 0.135"):* **MS135**

3. The file name may include other characters after the thickness number, to indicate a resonator type or processing application.

4. After the thickness number is usually chemical abbreviation for the cutting assist gas:

   **$O_2$** = Oxygen
   **$N_2$**= Nitrogen
   **AIR** = Air

When **G89** loads Process Parameters from file, the parameters can be edited while the program is running; however, changes may not take effect until the next **G84** (or **G85**). See the **PROCESS PARAMETER WINDOW** topic for more on viewing/editing Process Parameter files.

### G89 Using Command Line Arguments

Some Process Parameters can be set or modified directly using command line arguments to **G89**. This feature may be used for testing or other special cases where it is desired to temporarily set one or more parameters without editing a Process Parameter file. The following arguments can be used with **G89**:

**G89 T_ A_ I_ M_ S_ C_ D_ Q_ B_ E_ H_ R_ J_ K_ U_ V_**

**T** = Cut power level, watts.

**A** = Cut gas code. See Note 1.

**I** = Cut gas pressure. See Note 2.

**M** = Cut laser mode, see Note 3.

**S** = Cut pulse code, see Note 4.

**D** = Pierce time, seconds.

**Q** = Pierce power level, watts.

**B** = Pierce gas code. See Note 1.

**E** = Pierce gas pressure. See Note 2.

**H** = Pierce laser mode, see Note 3.

**R** = Pierce pulse code, see Note 4.

**J** = Pierce coolant code. See Note 5.

**K** = Kerf width, see Note 2.

**U** = Maximum feedrate for Dynamic Power, see Note 2.

**V** = Minimum percent for Dynamic Power (% at zero feedrate)

*Notes:*

1. Assist gas codes (**A** and **B**):

   **11** = Gas Port #1 (usually O2)

   **12** = Gas Port #2 (usually N2)

2. **G89** interprets pressures, kerf width, and dynamic power feedrate in the active units:

| Parameter | G20 unit | G21 unit |
|---|---|---|
| I and E | PSI | kPa |
| K | inches | mm |
| U | IPM | mm/min |

3. Laser mode (**M** and **H**):

   **61** = Continuous Wave

   **62** = Gated Pulse

   **66** = Dynamic Power

4. Pulse codes (**S** and **R**):

   When pulsed laser output is used, frequency and duty cycle are specified with a 4-digit code in which the first two digits specify frequency (Hz/100) and the last two digits specify duty cycle (%).

   *Note: For Laser Systems with a resonator, the maximum frequency is 2000 Hz and minimum duty cycle is the value necessary for a pulse on time of 100 microseconds at the commanded frequency.*

## G102 SET ADDITIONAL PROCESS PARAMETERS

**G102 A__ B__ S__** …etc... Set additional specified parameters

**G102** is used the same way as **G89** with command line arguments, to directly set or modify certain Process Parameters without editing a Process Parameter file. The following arguments can be used with **G102** to set Process Parameters that can not be directly set with **G89** arguments:

**G102 A_ B_ S_ Z_ D_ I_ T_ Q_ R_ U_ V_**

**A** = Dynamic gas pressure, Near field
**B** = Dynamic gas pressure, Far field
**S** = Pierce standoff
**Z** = Cut standoff
**D** = Precut dwell, seconds
**I** = Optional pressure
**T** = Power burst time, seconds
**Q** = Pierce Focus, Near field
**R** = Pierce Focus, Far field
**U** = Cut Focus, Near field
**V** = Cut Focus, Far field

**G102** interprets pressure, standoff and focus settings in the active units:

| Parameter | G20 unit | G21 unit |
|---|---|---|
| A, B and I | PSI. | KPa |
| S and Z | inches | mm |
| S, R, U and V | inches | mm |

## G103 SET RAMPED PIERCE PARAMETERS

**G103 A__ B__ C__** …etc... Set ramped pierce Process Parameters

**G103** can be used to directly set or modify ramped pierce power parameters without editing a Process Parameter file. The following arguments can be used with **G103** to modify the first five steps in a ramped pierce power sequence:

**G103 A_ B_ C_ D_ E_ F_ Q_ R_ S_ T_ U_ V_ W_**

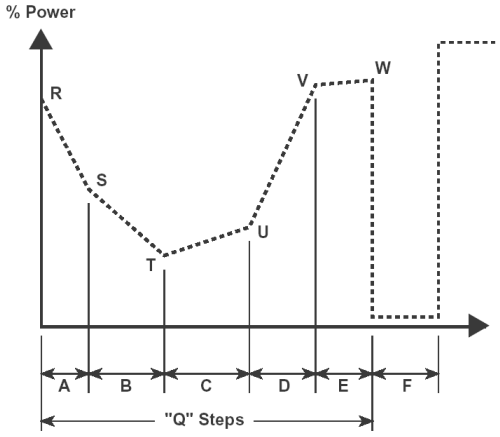
**A** = Ramp 1 duration, seconds
**B** = Ramp 2 duration, seconds
**C** = Ramp 3 duration, seconds
**D** = Ramp 4 duration, seconds
**E** = Ramp 5 duration, seconds
**F** = Tip cooling time, seconds
**Q** = Number of ramp steps (1 to 5)
**R** = Percent power at start of first ramp
**S** = Percent power at start of second ramp

**T** = Percent power at start of third ramp

**U** = Percent power at start of fourth ramp

**V** = Percent power at start of fifth ramp

**W** = Percent power at end of fifth ramp




## G120 AND G121 NON-STOP CUTTING

**G120** Disable Non-Stop Cutting
**G121** Enable Non-Stop Cutting

When a program commands Non-Stop Cutting (**G121**), the CNC replaces short **G00** moves between cut sequences with Smart Rapid moves. A Smart Rapid move commands the laser beam off and on without stopping the axes. (See "Smart Rapids" description.) During a Smart Rapid move, the control maintains assist gas flow, even when the laser beam is off.

*Notes: A program can only command Non-Stop cutting mode when the Process Parameters specify no pierce time and no precut dwell.*

*If the operator edits and saves Process Parameters while running a program in Non-Stop cutting mode, the CNC ignores the changes until the program ends.*

The **G120** command cancels Non-Stop cutting mode. When each program starts, the default mode is **G120**.

### Smart Rapids

Programming software normally commands a single **G00** linear rapid move between the end of one cut sequence and the beginning of the next. The **G00** move commands the shortest distance between the two points (to minimize the time between cuts). However, in the default mode (**G120**), the cutting head must stop before and after the **G00** move (to turn the beam off and on). In Non-Stop cutting mode (**G121**), the nozzle does not stop to turn the beam off or on.

When the operator loads a program, the control translates the program into commands for the CNC to execute. If the program specifies **G121** mode, the control translates **G00** moves into "Smart Rapid" moves. The control replaces the **G00** move and the contouring moves before and after it with commands that maintain the original beam-on path without stopping the axes.

The CINCINNATI laser control software performs these tasks to create a Smart Rapid:

1. Command **M35**.
2. Command a **G01** move after **M35** at the same feedrate and direction as the move before **M35**.
3. Command a **G01** move at high feedrate between the **G01** moves inserted in Steps 2 and 4.
4. Command a **G01** move before **G84** at the same feedrate and direction as the move after **G84**.
5. Command **G84**.
6. Lengthen the move after **G84**.

Although the CNC uses a linear **G01** connecting move (instead of **G00**), the high **G01** feedrate produces a curved path as the axes blend with the other **G01** moves. The result is a smooth non-stop transition between cuts.

If the original program has anything other than a single **G00** move between cuts (between **M35** and **G84**), the control does not create a Smart Rapid.

During a Smart Rapid move, the nozzle may deviate from the original programmed path while the beam is off. If the nozzle must follow the original path, then the program should not command **G00**. The control will not replace **G01, G02,** or **G03** moves with Smart Rapids.

*Note: In this description,* **G85** *can replace* **G84**.

*When the move before* **M35** *or after* **G84** *is an arc, the control inserts a Smart Rapid* **G01** *move tangent to the arc at the intersection point.*

### Beam On / Off Timing

The Laser System can turn the beam on and off at a desired time within 3 to 5 milliseconds. This means that the cut actually starts or stops within a distance representing 3 to 5 milliseconds of travel on either side of the desired point.

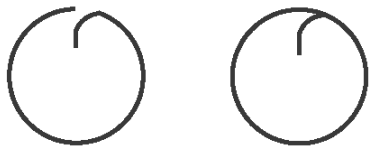
*Example:* At 300 inches/min., the length of the tolerance band could be (300 in./min.) / (60 sec./min.) * .005 sec. = ±.025 in. (0.63 mm)

*Note: The size of the tolerance band depends on the control design. For machines built prior to July 1999, the tolerance is ± 5 milliseconds. For all other Laser Systems, the tolerance is ± 3 milliseconds.*

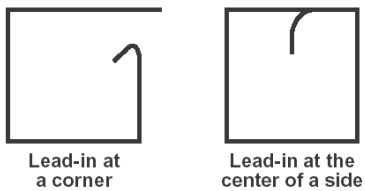
### Cut Extension

The CNC determines the **G121** extension time to control how the Laser System cuts shapes in **G121** mode.

Consider a round hole with a radial lead-in. In conventional cutting, the programmer might end the lead-in with exact stop (**G09**). However, a lead-in programmed for Non-Stop mode may not command exact stop, and the end of the lead-in would blend into the start of the circular move. Also, beam On/Off positions are not as precise when using Smart Rapids (as described above). Thus, the slug from a 360-degree circular hole may not drop because the beam would not cut the entire perimeter of the hole (see figure). Extending the circular move assures that the beam will cut the entire shape.

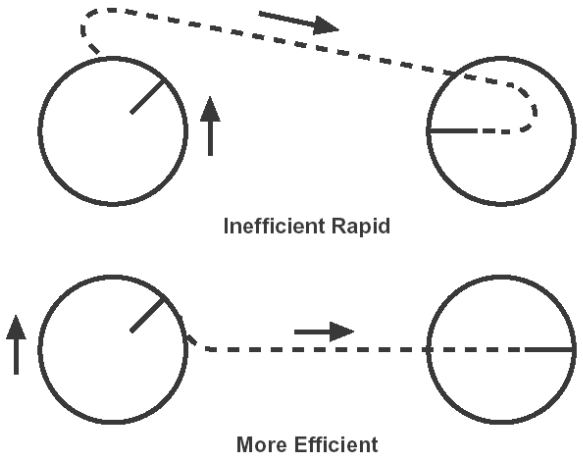



It is important to place the lead-in so the extension of its last entity will meet or overlap the beginning of the first contoured entity. If the program commands a lead-in at a corner, the extension of the last entity will cut past the desired perimeter of the feature, as shown in the following diagram.



### Efficiency

As described above, the path of a Smart Rapid is not a straight line. The ends are smoothed, the way a baseball player rounds the base paths on an extra base hit. Program efficiency is greatest when the path length is shorter. Path length between two features depends on cutting direction and placement of lead-ins, as shown in the following diagram.




The cases shown in the figure have two programming differences: the cutting direction of the left hole, and the location of the lead-in on the right hole. Note the "U-turns" required in the top path. When selecting lead-in locations, the desire to minimize Smart Rapid path length may conflict with the desire to maximize "head down" operation (by avoiding tipped slugs). The conflicting requirements often require some compromise.

## G123, G124, AND G125 VELOCITY BLENDING

| | |
|---|---|
| **G123 A__ S__** | Programmable Blend |
| **G124** | Use Default Blending |
| **G125** | Use Auto Blending |

The process of ending one contouring move and beginning the next move usually requires changing the velocity of one or both axes. In this manual, this process is called a "blend".

The objective of a blend is to change the axis from executing the preceding move at its constant velocity to executing the next move at a different constant velocity. To accomplish this transition, acceleration also changes during the blend. When velocity is constant, acceleration is zero. Therefore, during a blend (between linear moves, for example), acceleration begins at zero, increases or decreases to produce the velocity change, then returns to zero to complete the blend.

The CINCINNATI laser control executes a blend using two parameters: the overall time to complete the velocity change and the portion of that time which is used to change the acceleration. Just as a motion control system has a maximum velocity and a maximum acceleration, it is also limited by how quickly it can change acceleration.

Short blend times improve contouring accuracy and increase productivity by using high acceleration, but can produce servo following errors if the machine attempts to exceed its acceleration capability. Long blend times avoid servo following errors but sacrifice contouring accuracy. Since blend time settings can affect processing results, users can program blend times with three different commands:

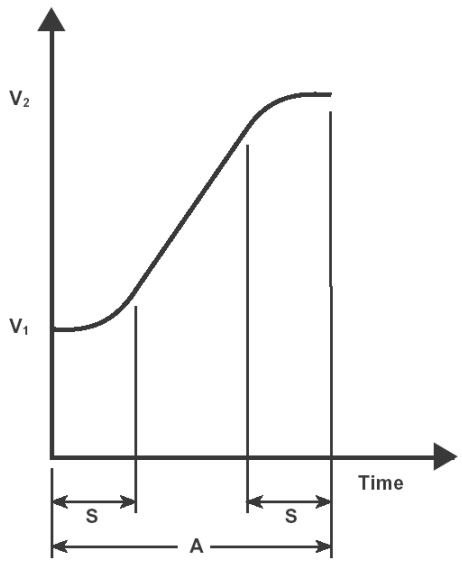
**G123** specifies fixed time values for all blends:

**G123 A_ S_**

**A** = Total blend time, in milliseconds

**S** = Time for acceleration change, in milliseconds

The minimum value of **A** is **S** * 2.




**G124** commands the control to use a set of default times for all blends. The default time values are set by CINCINNATI INCORPORATED.

**G125** Auto Blend enables the control to determine the minimum *X and Y Blend times* independently for each move. When each program starts, the default mode is **G125**.

Different parts of a program can use different blend control modes. A program can change from **G123**, **G124**, or **G125** to either of the other modes.

If either **G123** or **G124** setting is less than a minimum time, the control uses the minimum time without displaying an error message. The control determines the minimum blend time for each move using parameters set by CINCINNATI INCORPORATED.

# M CODES

M codes are used for program control of specific machine functions, usually unrelated to x/y axis motion. See the **M Code List** for a table of all M codes supported by CINCINNATI Laser System controls.

## M00 PROGRAM STOP

**M00** Program Stop

When the CNC executes a program block commanding **M00** (or **M0**), the program stops until the operator presses CYCLE START. If the laser beam and assist gas were on, the **M00** command turns them off. **M00** places the CNC in a Cycle Stop condition. Modal information does not change.

## M01 OPTIONAL STOP

**M01** Program Stop (only if Optional Stop mode is active)

The **M01** (or **M1**) command has the same function as the **M00** command, except **M01** is only executed when Optional Stop Mode is active. Use the "Optional Stop" button on Program Modes Control Bar to enable/disable Optional Stop mode. **M01** is ignored if Optional Stop mode is disabled.

## M02 END OF PROGRAM

**M02** End of program

A CNC program can use **M02** (or **M2**) as the last block. This function disables all processing functions, resets all previously requested M codes and prevents further execution. The program does not rewind automatically after **M02** is executed. When the operator loads a program into the CNC, the control ignores any codes following **M02**.

## M30 END OF PROGRAM / REWIND

**M30** End of program with rewind

Most CNC main programs use **M30** as the last block. This function disables all processing functions, resets the CNC, cancels all previously requested M codes, prevents further execution and rewinds the program.

## M35 BEAM OFF

**M35** Turn laser beam off

**M35** turns the laser beam off at the end of a cut sequence. Laser discharge current stops and remains off until the next **G84** or **G85** is executed. Assist gas flow also stops, unless the Speed Gas option in the *Variables* menu is selected (see **M135**). **M35** dot not cause the shutter to close.

## M36 Z HOLD MODE

**M36** Activate Z-axis Servo Hold mode

The **M36** command places the Z-axis control for the noncontact head in a servo hold condition when the nozzle is in the cutting position and the laser beam is on. The noncontact head does not follow the material after the program commands **M36**.

Once **M36** is executed, Z Hold mode remains active until **M35**, **M37**, **M42**, **M47**, or **M30** is executed, or the "RESET" button is pressed.

## M37 BEAM AND GAS OFF / SHUTTER CLOSE

**M37** Turn laser beam and assist gas off, and close the shutter

**M37** turns off the laser beam and assist gas and commands the shutter to close. **M37** also resets any previously requested M codes.

To avoid unnecessary cycling of the gas valves and shutter, most programs use **M35** instead of **M37**. The control accepts **M37** to support programs written for Laser Systems without **M35**.

## M38 Z HOLD MODE (TIMED)

**M38** **P__** Activate Z-axis Servo Hold mode for a specified time

**M38** places the Z-axis control for the noncontact head in a servo hold condition for a period of time after the control establishes the cutting position. The **M38** block specifies the time with the **P** argument (in milliseconds). The valid range for **P** argument values is 0 to 2 minutes (**P0** to **P120,000**).

*Example: Programming the Z-axis to maintain a fixed position for 2 seconds after reaching the nozzle standoff position:*

**M38** **P2000**

To start the **M38** time, the Z-axis must be in the cutting position with the beam on, and the X-axis or Y-axis must be moving. The Z-axis maintains position until the time has elapsed. After the specified time, the Z-axis goes into tracking mode.

The program can command **M38** before or after **G84** or **G85**. If **M35**, **M37**, **M42**, **M47**, or **M30** is executed, or the "RESET" button is pressed before the specified time elapses, Servo Hold mode activated by **M38** will be canceled.

## M41 Z DOWN

**M41** Move Z-axis down to cut position

**M41** commands the Z-axis to move the cutting nozzle down to the commanded standoff position. Programs normally do not use **M41** because **G84** and **G85** include that function. **M41** allows a program to command the cutting nozzle down to the material without starting a cut.

If the noncontact cutting head does not detect material, the Z-axis travels down until it reaches a minimum position or exceeds a time limit, causing motion to stop.

## M42 Z UP

**M42** Retract Z-axis to home position

**M42** commands the Z-axis to the full up (home) position. The shutter is commanded to close. Programs use **M42** to command the Z-axis to the required position for pallet motion.

## M43 LOWER PALLET SPECIAL FUNCTION

**M43** Activate Lower Pallet Special Function mode

In normal operation, the control only allows the beam to turn on when the cutting nozzle is within 1.5 inches (38mm) of the material support height on either pallet. On Laser Systems equipped with the Lower Pallet Special Function option, users can extend the allowable beam-on range to 7.0 inches (178mm) on the lower pallet by commanding **M43**.

The **M43** Lower Pallet Special Function operating mode allows the machine to process square or rectangular tubing or other formed parts on the lower pallet. Operation above 1.5 inches (38mm) is not possible on the upper pallet because the Z-axis reaches its upper limit.

**M43** changes these control functions:

1. The Z-axis travels down from the top position at a lower speed to find the material surface.
2. The pallets cannot be moved with **M50** or with pallet "Jog" buttons.
3. The Y-axis cannot exceed the machine coordinate specified on the CONFIGURATION window for this option.

**M43** disables pallet motion because material over 1 in. (25mm) high will not clear the upper pallet. Since the pallets cannot move and the X-axis beam delivery blocks access to the pallet from one side of the main frame, the operator must load and unload the pallet from the other side of the main frame (the operator side). To keep material within reach of the operator side, the control does not allow the Y-axis to exceed the machine position set on the CONFIGURATION screen.

Recommended procedure to use **M43**:

1. Move the upper pallet OUT and the lower pallet IN.
2. Jog the nozzle to machine Y0.
3. Load and execute a program with only **M43** and **M30**.
   (CINCINNATI provides such a program named "M43.cnc".)
4. Verify **M43** mode is active (FYI message).
5. Load material on the lower pallet.
6. Load the CNC program for the loaded material.

Programs using **M43** mode can begin with **M43**, but to avoid accidental damage, do not load material over 1 in. (25mm) high until **M43** mode is already active. The control will execute the **M43** command with Program Test mode on or off.

**CAUTION**

**Do not load material over 1 inch (25 mm) high on the lower pallet until M43 mode is active. When material over 1 inch high is on the lower pallet and M43 mode is not active, operation in JOG or AUTO mode can damage the machine and the workpiece.**

When a Laser System has this option, the CNC is in **M43** mode each time the operator turns on the machine control. **M43** mode is NOT canceled by **M30**, RESET, or turning off the control. The only way to cancel **M43** mode is to run a program commanding **M44**.

## M44 DISABLE LOWER PALLET SPECIAL FUNCTION

**M44** Disable Lower Pallet Special Function mode

**M44** cancels the Lower Pallet Special Function mode commanded with **M43**. The **M44** command restores normal Z-axis speed, pallet motion and Y-axis travel range. The control will execute the **M44** command with Program Test mode on or off.

To cancel **M43** mode:

1. Remove any material or fixture over 1 in. (25 mm) high.
2. Load and execute any program beginning with **M44**.

**CAUTION**

**Remove any workpiece or fixture over 1 inch (25 mm) high before executing a program with M44. When material over 1 inch high is on the lower pallet and M43 mode is not active, operation in JOG or AUTO mode can damage the machine and the workpiece.**

## M45 OPTIONAL STANDOFF MODE

**M45** Apply Optional Standoff For Cutting

When the CNC program commands **M45** after starting a cut sequence, the noncontact head standoff changes to the "Optional Standoff" distance specified in the active Process Parameter file. The **M45** command does not change pierce standoff. If the head is not already tracking material, an **M41** or **G84** will still be necessary to command the head down.

## M47 PARTIAL Z UP

**M47 (P__)** Raise Z-axis, Optionally by Distance

Programs can partially raise the Z-axis with **M47,** to avoid interference with clamps or tipped parts during non-cutting moves.

**M47** commands the Z-axis to a preset, "partially up" distance (as opposed to the "completely up" home position as with **M42**). The default partial up distance is the user-configurable **Default Partial Z-up Distance (M47)** parameter in the *Z-axis Setup* window.

The user can override the default partial up distance on a per-block basis by using the **P** argument with **M47.** Use **M47 P** distance to program a specific Z Up distance for individual features. The distance units are thousandths of an inch for **G20** mode and thousandths of a millimeter in **G21** mode. The valid range for **P** argument values is 0 to 10 inches (**P0** to **P10000**), or 0 to 254 mm. (**P0** to **P254000**) if **G21** mode is active.

## M48 FEEDRATE OVERRIDE DISABLE

**M48** Inhibit the **Feedrate Override** function

**M48** disables the feedrate override setting specified by the Override Adjust Control Bar and sets the feedrate to 100% of the value specified in the program. **M48** is canceled by **M49**, **M30,** or the "RESET" button.

## M49 FEEDRATE OVERRIDE ENABLE

**M49** Enable the **Feedrate Override** function

**M49** restores the function of the feedrate override specified by the Override Adjust Control Bar . When a program starts, the default mode is **M49**. This function cancels **M48**.

## M50 SWITCH PALLETS

**M50** Switch positions of the upper and lower pallets

**M50** commands the upper and lower pallets to switch positions. The pallets will switch positions only if the "PALLET NOT READY" pushbutton/indicator is not illuminated. If necessary, the **M50** command will also retract the cutting head (like **M42**).

The control illuminates the "PALLET NOT READY" pushbutton when a program starts. The illuminated button indicates the pallets are "Not Ready" to switch positions. Press the button to toggle the status on or off before the program reaches the **M50** block. If the button is not illuminated when the program executes an **M50** block, the pallets will switch positions.

If the "PALLET NOT READY" button is illuminated when the program reaches the **M50** block, the program will stop. To resume the program press the "PALLET NOT READY" button and the pallets will reverse positions.

## M51 AUXILIARY OUTPUT (TIMED)

**M51 P__** Turn auxiliary output on for a specified time

**M51** commands a set of isolated relay contacts to close for the number of milliseconds specified by the **P** argument. The auxiliary output contacts can be used for customer-specific interface functions. The valid range for **P** argument values is 0 to 10 seconds (**P0** to **P10000**). The default time is 0.5 seconds will be applied if no **P** argument is included in the **M51** block.

*Example: Commanding the auxiliary contacts to close for one second:*

**M51 P1000**

## M67 OPTIONAL ASSIST GAS PRESSURE

**M67** Apply Optional Assist Gas Pressure

**M67** changes the cutting assist gas pressure command to the value specified by the **Pressure 2 (Optional, M67)** setting in the most recently-executed Process Parameter file, or as specified by **G102 I__**. The pressure command is valid until replaced by the next **G84** or **G85**.

## M90 BALL TRANSFER UP

**M90** Raise the ball transfer mechanism to the "UP" position

**M90** commands the optional ball transfer mechanism to the "UP" position. This feature is typically used in conjunction with automated material handling systems, such as the CINCINNATI MMHS.

## M91 BALL TRANSFER DOWN

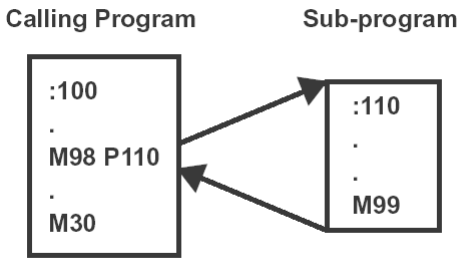
**M91** Lower the ball transfer mechanism to the "DOWN" position

**M91** commands the optional ball transfer mechanism to the "DOWN" position. This feature is typically used in conjunction with automated material handling systems, such as the CINCINNATI MMHS.

## M98 SUBPROGRAM CALL

**M98 P__ (L__)** Subprogram call with no arguments

The **M98** command transfers control from the calling program to a subprogram. The **M98** block specifies the subprogram name after **P**. If the subprogram is in the same file as the calling program, only the program name is required. If the subprogram is in a separate file, **P** is followed by the file name including its extension (if any) and its path if different from the calling program.




The program can call the subprogram more than once by specifying the number of times with **L**.

*Example: Calling a subprogram named "1200" three times:*

**M98 P1200 L3**

When a program calls a subprogram with **M98**, the two programs share the same set of local variables (See the **SUBPROGRAMS AND MACROS** topic).

## M99 END SUBPROGRAM

**M99 (P__)** End a subprogram and return control to the main program

The **M99** command returns control to the program that called the subprogram. The block following the subprogram call is executed next. Subprograms called with **M98** or **G65** can end with **M99.**

If a main program commands **M99**, the control restarts the main program from the beginning.

If the **M99** command includes the optional **P** argument, the subprogram returns to the calling program at the sequence number specified after **P**. If the **M99 P** command is in the main program, the control returns to the line number specified by **P** in the same program (same as **GOTO**).

*Example:*

**M99 P500**

(If commanded in a subprogram, this example returns to the calling program at line **N500**. If commanded in a main program, this example returns to line **N500** in the main program.)

## M130 AND M131 Z-AXIS ANTIDIVE

M130 Disable Z-axis Antidive

**M131** Enable Z-axis Antidive

**M130** and **M131** disable and enable the **Z-axis Antidive** function. **Z-axis Antidive** is a feature that can help avoid damage to the cutting head due to programming errors.

**M131** enables the **Antidive** function. The cutting head follows limited variation in the material surface but maintains Z-axis position when it does not detect material. **M131** is the default mode. All programs start in **M131** mode, with **Antidive** enabled.

**M130** disables the **Antidive** function. If **M130** is active, the Z-axis does not use Antidive mode. The Z-axis lowers the cutting head until it detects material or an overtravel alarm occurs.

When a program commands **M130**, the control disables Antidive until one of the following occurs:

1. The program commands **M131**.
2. The program commands **M30**.
3. The operator rewinds the program.
4. The operator loads a new program.

While **M130** is active, the control displays the FYI message: *"Z-axis antidive is disabled."*

Programs use **M130** for applications with significant material vibration. **M130** allows the head to follow moderately warped material or thin gage material that flutters due to interaction with assist gas pressure.

**CAUTION**

**Programmers are advised to carefully consider the application before using M130. If the program commands the cutting head to travel over a hole in M130 mode, the cutting head will most likely dive into the hole and breakaway at the magnet flange, possibly damaging the cutting head.**

## M135 SPEED GAS BEAM OFF

**M135** Turn laser beam off; leave assist gas on

**M135** is similar to **M35** except **M135** leaves the assist gas on. Discharge current remains off until the next **G84** or **G85**. **M35** acts like **M135** when the Speed Gas option in the *Variables* menu is selected. **M135** leaves the gas on independent of the SPEED GAS selection.

## M151 AUXILIARY OUTPUT WITH CONFIRMATION

**M151** Turn auxiliary output on and wait for confirmation input

Similar to **M51**, **M151** causes the auxiliary output relay contacts to close. In addition to activating the auxiliary output, **M151** causes NC program execution to be suspended until the corresponding **M151** customer input is received by the laser control.

The **Auxiliary Output (M151) Time-out** parameter in the Auxiliary Configuration window sets a limit on how long the laser control will wait for the confirmation input. If the **M151** auxiliary input is not toggled on/off in less than this length of time, a system alarm, *"Auxiliary input (**M151**) did not turn on"*, will be reported, putting the laser control in the Cycle Stop state.

# CINCINNATI MACROS

CINCINNATI macro programs simplify programming for common applications. The macros are in two groups: grid macros and cutting macros. Grid macros call a user's subprogram in a rectangular pattern of rows and columns. Cutting macros cut common shapes based on specified dimensions.

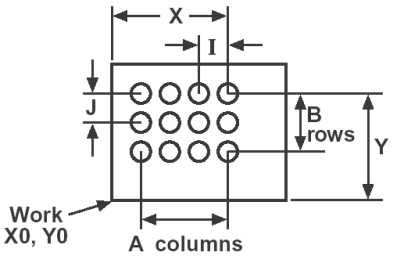
## GRID MACROS

Programs use CINCINNATI grid macros to repeat a subprogram in a pattern of rows and columns. The subprogram can repeat a feature within a part, or repeat a part on a sheet.

### 9800 Part Sub Grid Macro

**G65 P9800 A_ B_ I_ J_ X_ Y_ S_ (R_ K_)** Repeat part feature in a rectangular grid pattern

Programs can use the **P9800** grid macro to repeat a part feature in a rectangular grid pattern. A typical application is a part with an array of holes or slots. The user must provide a separate subprogram to cut one feature. The part program calls the grid macro once, and the grid macro calls the subprogram several times to cut the features.




***In this figure, the grid macro calls the subprogram at the center of each hole.***

**A** = Number of subprogram calls in local x direction (columns).

**B** = Number of subprogram calls in local y direction (rows).

**I , J** = Local x and y distances between subprogram calls.

**X, Y** = Local x and y coordinates where the grid macro calls the subprogram farthest from local X0, Y0.

**S** = Subprogram name.
Since the macro call provides the subprogram name as a macro argument, the name must be an integer number with no extension. The subprogram can be in the same file as the calling program or a separate file in the same directory. If the subprogram is a separate file and the file name has an extension, the grid macro will not find the subprogram. To rename the file without an extension, use Windows Explorer.

**R** = Rotation angle for the subprogram relative to the part coordinate system, in degrees.
Default angle is zero with counterclockwise positive.

**K** = Quantity of subprogram calls for the grid macro to skip. Default is zero.
When restarting an interrupted program, the operator can use the K argument to make the macro skip some of the subprogram calls. The macro skips the number of calls specified by K and begins with the subprogram call at the next position.

### G65 P9800 Description

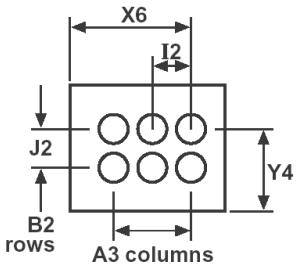
The **9800** grid macro moves the cutting head in rapid traverse to the locations defined by **A, B, I, J, X,** and **Y**; and calls the subprogram from each location. When the macro call specifies **R**, the macro commands coordinate rotation before calling the subprogram. If the subprogram is written is **G91** mode, it must end at its starting point.

The **9800** grid macro does not raise or lower the cutting head. The macro maintains the Z-axis position at the end of the subprogram for the move between subprogram calls.

The grid macro calls the first subprogram from work coordinates X, Y when **K** is zero. To complete the first row, the macro proceeds in the local -x direction. The second row begins under the first part at a lower y coordinate, and repeats in the local -x direction. This procedure continues until the macro completes all rows.

*Example Program Using P9800:*

This example part has six holes 1.5 in. diameter, on 2 in. centers in three columns and two rows in the center of an 8 x 6 rectangle.




**O2000** (PART PROGRAM WITH 9800)

```
G20 G90 F100
G89 PMS135.LIB
G92 X0 Y0
G65 P9800 A3 B2 I2 J2 X6 Y4 S2001 R0 K0
G86 X4 Y3 I8 J6
M30
```

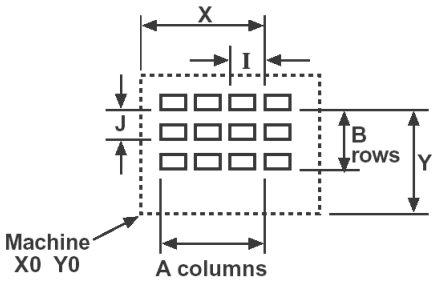
**O2001** (HOLE SUBPROGRAM)

```
G91
G73 X0 Y0 D1.5
G90
M99
```

### 9900 Part Grid Macro

**G65 P9900 A_ B_ I_ J_ X_ Y_ S_ (R_ K_ Z_)** Repeat part in rectangular grid pattern

Programs use the 9900 grid macro to repeat a part in a rectangular grid pattern on the sheet. The program calls the grid macro once and the grid macro repeatedly calls a subprogram for one part.




***In this figure, the grid macro calls the subprogram from the lower left corner of each part.***

**A** = Number of parts in machine x direction (columns).

**B** = Number of parts in machine y direction (rows).

**I, J** = Distances between part calls in machine x and y directions.

**X, Y =** Machine coordinates where the grid macro calls the part farthest from machine X0, Y0.

**S =** Part program name. Since the macro call provides the part subprogram name as a macro argument, the name must be an integer number with no extension. The subprogram can be in the same file as the calling program or a separate file in the same directory. If the subprogram is a separate file and the file name has an extension, the grid macro will not find the subprogram. To rename the file without an extension, use Windows Explorer.

**R =** Rotation angle for the part program (relative to the machine coordinate system), in degrees. Default angle is zero with counterclockwise positive.

**K =** Quantity of parts to be skipped before the first is cut. Default is zero. When restarting an interrupted program, the operator can use the **K** argument to make the macro skip some of the subprogram calls. The macro skips the number of calls specified by **K** and begins with the subprogram call at the next position.

**Z =** Z-axis flag. Default is **Z** (to raise head between parts). When the macro call does not specify **Z**, or specifies **Z** with zero or a positive number, the grid macro commands **M47** to raise the cutting head before moving into position to call the subprogram for each part. When the macro call specifies **Z-1** (or any negative value), the macro still commands **M47** before the move to the first part, but does not command **M47** between parts in the same row or the same column. The macro commands **M47** between the last part in one row and the first part in the next row when the grid has more than one part per row (**A>1**).

### G65 P9900 Description

The **9900** grid macro moves the cutting head in rapid traverse to the locations defined by **A**, **B**, **I**, **J**, **X**, and **Y**, and calls the part subprogram from each location.

The macro calls the first subprogram from machine coordinates X, Y when **K** is zero. To complete the first row, the macro proceeds in the -x machine direction. The second row begins under the first part at a lower y coordinate, and repeats in the machine -x direction. The macro continues this procedure until it completes all rows.

When the macro returns to the calling program, it maintains the cutting head Z position commanded at the end of the last subprogram.

When the macro call specifies **R**, the macro commands coordinate rotation before calling the subprogram. The macro rotates the part about its starting point. Therefore, the programmer must consider the resulting position of the starting point when determining the **I**, **J**, **X**, and **Y** values.

The grid macro commands **G92 X0 Y0** before rotating the coordinate system. After a main program calls the grid macro with **R** > 0, the main program should use **G53** to move between parts, or re-establish the work coordinate system with **G92**.

To avoid an extra move at the start of each part, the subprogram should begin with the cutting head already at the coordinates of the first pierce. If programmed in absolute (**G90**) mode, local X0, Y0 should be the location of the first pierce.

*Example Program Using P9900:*

This program calls subprogram **1001** to cut a part 24 times in a pattern with 6 columns and 4 rows. The macro calls the first part first part from machine X48, Y24 and the part spacing is 6 inches in x and 4 inches in y.

```
O1000 (MAIN PART GRID WITH 9900)
    G20 G90 F100
    G89 PMS135.LIB
    G65 P9900 A6 B4 I6 J4 X48 Y24 S1001 Z1
    M30
O1001 (PART SUBPROGRAM)
    G92 X0 Y0
```

```
G0 X_ Y_
G84
.
.
M35
M99
```

## CUTTING MACROS

The cutting macros internally call **G84** to begin each cut. **G79** can also call **G85**. **G84** or **G85** then uses the cutting parameters specified by **G89**, **G102** and **G103**. The user's program calls each macro with a series of arguments to specify programming options. Optional arguments are shown in parentheses.

| | |
|---|---|
| **G73** | Hole |
| **G76** | Slot |
| **G79** | Line |
| **G83** | Outside Circle |
| **G86** | Outside Rectangle |
| **G88** | Bolt Circle |
| **G104** | Shape |
| **G105** | Lead-in |

The **X** and **Y** arguments for **G79** are machine coordinates. For the other cutting macros, **X** and **Y** are absolute work coordinates when called in **G90** mode, and incremental distances when called in **G91** mode.

The only cutting macro that raises the head before its first move is **G79.** The others rely on the cutting head position (established by the calling program before the macro call) to avoid interference from tipped slugs or clamps.

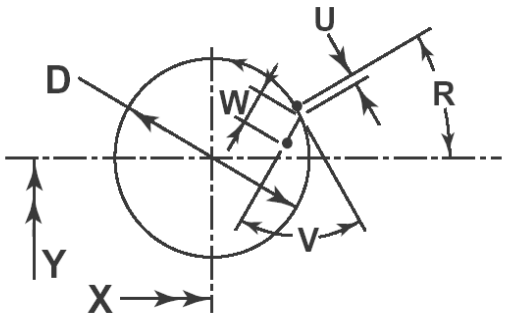
Except for **G79**, the cutting macros automatically apply kerf compensation (set by **G89 K_** or the parameter library file).

**G79** returns from the end of the cut and **G105** returns from the end of the lead-in. The others return from the end of the cut when called in **G90** mode, or from the center of the contoured shape when called in **G91** mode.

Before returning to the calling program, **G79** always raises the cutting head. **G105** returns to the calling program with the head down and the beam on. The user can program the other macros to either raise the head or leave it down when returning to the calling program.

### G73 HOLE MACRO

**G73 X_ Y_ D_ (A_ B_ F_ H_ K_ M_ Q_ R_ T_ U_ V_ W_ Z_)** Hole Macro

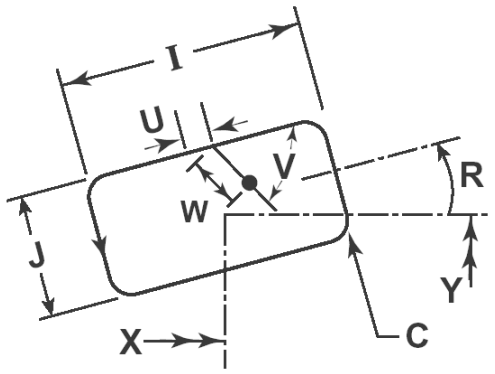



**X, Y** = Hole center **X**, Y coordinates
**D** = Hole diameter

**Others** = Optional arguments, see **G104**

## G76 SLOT MACRO

**G76 X_ Y_ I_ J_ (A_ B_ C_ F_ H_ K_ M_ Q_ R_ T_ U_ V_ W_ Z_)** Slot Macro




**X, Y** = Slot center **X, Y** coordinates

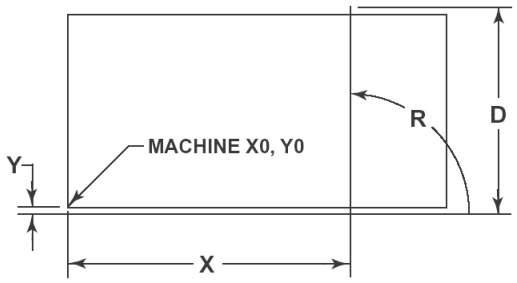
**I, J** = Slot overall dimensions in local x and y directions (as if **R** = 0)

**C** = Corner radius, default = 0

**Others** = Optional arguments, see **G104**

## G79 LINE MACRO

**G79 X_ Y_ D_ R_ (E_ H_)** Line Macro




**X, Y** = Machine coordinates at start of cut

**D** = Length (distance) of cut

**R** = Rotation angle for cut direction, in degrees.

*Examples:*

**R**0 = Machine +**X** direction
**R**90 = Machine +**Y** direction
**R**180 = Machine -**X** direction
**R**-90 = Machine -**Y** direction

**E** = Edge start flag:
**E**0 uses **G84** (default).
**E**1 uses **G85** (no pierce).

**H** = Optional pressure flag, see **G104**.

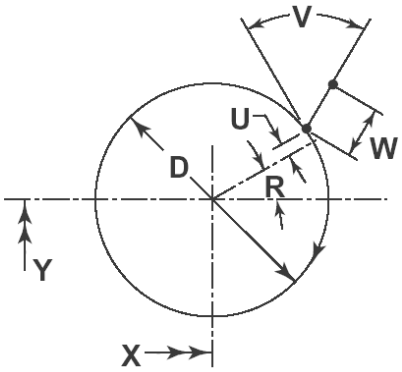
### Description

A program can use **G79** to cut a sheet, usually to separate the skeleton from a usable remnant or to cut the skeleton into smaller pieces for easier removal. The **G79** macro raises the cutting head, moves to machine x, y in rapid traverse and begins the cut with **G84** or **G85**. **G79** then commands **M67** if **H1** is set, completes the cut at the program feedrate, ends the cut with **M35** and retracts the cutting head with **M42**.

*Note: **G79** does not use a lead-in and does not check for interference with sheet clamps or material stops.*

## G83 OUTSIDE CIRCLE MACRO

**G83 X_ Y_ D_ (A_ B_F_ H_ K_ M_ Q_ R_ T_ U_ V_ W_ Z_)** Outside Circle Macro



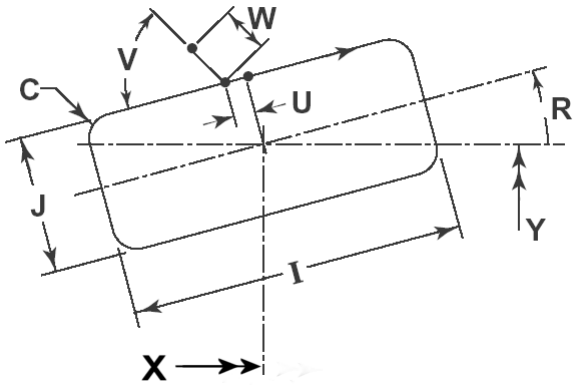

**X, Y** = Hole center **X**, **Y** coordinates

**D** = Hole diameter

**Others** = Optional Arguments, see **G104**.

## G86 OUTSIDE RECTANGLE MACRO

**G86 X_ Y_ I_ J_ (A_ B_ C_ F_ H_ K_ M_ Q_ R_ T_ U_ V_ W_ Z_)** Outside Rectangle Macro




**X, Y** = Rectangle center **X**, **Y** coordinates

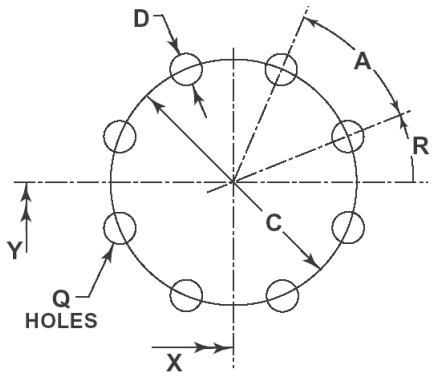
**I, J** = Rectangle overall dimensions in local x and y directions (as if **R** = 0)

**C** = Corner radius, default = 0

**Others** = Optional arguments, see **G104**.

## G88 BOLT CIRCLE MACRO

**G88 X_ Y_ C_ D_ Q_ (A_ B_ F_ H_ K_ M_ R_ T_ U_ V_ W_ Z_)** Bolt Circle Macro




**X, Y** = Bolt circle center x, y coordinates

**C** = Bolt circle diameter

**D** = Hole diameter

**Q** = Number of holes. Since **G88** uses **Q** for the number of holes, the hole macro called by **G88** cannot use **Q** for the number of segments after type **M2** lead-in (See **G104**). When a program calls **G88** with **M2**, the hole macro uses the default number of segments.

**A** = Angle between holes, in degrees. Default = [360 / **Q**]. The **G88** macro call uses **A** when the application only requires part of a complete bolt circle. Since **G88** uses **A** for this function, the hole macro called by **G88** cannot use **A** for the first dwell of type **M2** lead-in. When a program calls **G88** with **M2**, the hole macro uses the default first dwell.

**R** = Rotation angle from local +X-axis to first hole, in degrees. The default angle is zero. Counterclockwise is positive. Since **G88** uses **R** for rotation angle, the hole macro called by **G88** cannot use **R** for the lead-in position of each hole. The hole macro uses the default lead-in position for all holes.

**Others** = Optional arguments, see **G104**.

### Description

**G88** moves to the specified **X** and **Y** coordinates in rapid traverse, and then calls **G104** to cut each hole at its calculated coordinates. The macro raises the cutting head between holes if programmed by the **Z** argument (see **G104**). **G88** returns from the center of the last hole when called in **G90** mode, and from the center of the bolt circle when called in **G91** mode.

## G104 SHAPE MACRO

**G104 X_ Y_ D_ (A_ B_ E_ F_ H_ K_ M_ Q_ R_ T_ U_ V_ W_ Z_)** Circular Shape Macro

**G104 X_ Y_ I_ J_ (C_ A_ B_ E_ F_ H_ K_ M_ Q_ R_ T_ U_ V_ W_ Z_)** Rectangular Shape Macro

A program can use **G104** to cut circular or rectangular shapes as internal cutouts or part outlines. Since **G73**, **G76**, **G83** and **G86** call **G104** to produce their shapes, to avoid one level of subprogram nesting use **G104** instead of those macros.

**X, Y** = Shape center coordinates

**D** = Circle diameter

**I, J** = Rectangle dimensions. **I** = local x, **J** = local y. The macro interprets **I** and **J** as if **R** was zero.

**C** = Rectangle corner radius, default = 0

**E** = External cut flag:

**E0** = Internal (default)
**E1** = External (for part outlines)

**F** = Feedrate for lead-in. Minimum is 30 IPM or the program feedrate if lower.
Default and maximum depend on **M**:
For **M0**, **F** is the lead-in feedrate. Default is 30 IPM. Maximum is contouring feedrate.
For **M1**, **F** is the feedrate of the first move after the lead-in. Default and maximum is the program feedrate.
The macro commands the lead-in using three steps with feedrates of 20, 40 and 60 % of **F**.
The **U** move is at 80 % of **F**.
For **M2**, **F** is the feedrate of the final lead-in move (see **M2** description). Default and maximum is the contouring feedrate.

**H** = Optional pressure flag:
**H0** = Off (default)
**H1** = On. **G104** commands **M67** at the end of the lead-in when the macro call specifies **H1**.

**M** = Lead-in type:

| | | |
|---|---|---|
| **M0** | = | Single feedrate (default) |
| **M1** | = | Multi-step at increasing feedrates |
| **M2** | = | Cross lead-in followed by segments with increasing feedrates |

**R** = Shape rotation angle in degrees. Default is zero. Counterclockwise is positive.
For circles: R defines lead-in position, where default is local +x intersection.

*Note: Since **G88** uses **R** for the angle from **+x** to the first bolt hole, all **G88** bolt holes have the default lead-in position.*

**T** = Contouring accuracy tolerance. Default is .001 in. (.025 mm).

**G104** calculates the feedrate for arcs and circles based on radius and tolerance. The macro uses the calculated feedrate unless it exceeds the modal program feedrate. The programmer can use T to affect the calculated feedrate.

Since the CINCINNATI cutting macros use the **T** argument for radius tolerance, they cannot command pierce options with **T1**, **T2** and **T3** like **G84.** Instead, cutting macros can specify the pierce option with a decimal digit after the M argument. For example, to command a hole with a single entity lead-in (type **M0**) and pierce option 2, program: **G73 X_ Y_ D_ M0.2**

The default pierce option is 1. For example, (**G73**) **M2** is the same as **M2.0** or **M2.1**. Sheet cutoff macro **G79** uses the decimal digit of M for pierce option even though **G79** has no lead-in (**G79** ignores the ones digit of **M**).

**U** = Length of last lead-in entity:

For **M0** and **M1**: **U** is the length of the last lead-in move, which is commanded in line with the first contouring move. Default and minimum length is one kerf width.
For **M2**: **U** is the length of each contouring segment inserted after the lead-in. Default and minimum length is .080 inch.
For circles with **M2**: Maximum **U** = [π * **D**] / [4***Q**] (The arc segments are always completed in the first quadrant.)
For rectangles with **M2**: Maximum **U** is the distance from the center of the longer side to the start of the corner radius, divided by **Q**.

**V** = Angle between lead-in line and first contouring move, in degrees.
Default is 90 degrees (perpendicular lead-in).
The macro also uses the default angle when the call specifies **V0**. Use **V360** to command the lead-in in the same direction as the first contouring move.

**W** = Lead-in length.
For **M0** and **M1**: Default **W** is the smaller of .25 in. or half the minimum width of the shape.
For **M2**: Default **W** is one-fifth of the minimum width of the shape, but not more than 0.2 in.
Minimum **W** is eight kerf widths.

**Z** = Z-axis retract flag.
When the macro call does not include **Z**, or specifies **Z** with zero or any positive number, **G104** commands **M47** to raise the Z-axis after cutting the shape. To leave the cutting head on the material, call **G104** (or **G73** etc.) with **Z**-1.

**Arguments Associated with M2 Only**

**A** = Dwell (seconds) before lead-in move. Default = 0.250 seconds. The macro also uses the default dwell if the call specifies **A0**.

*Note: **G88** uses **A** for the angle between holes. When a program calls **G88** with **M2**, this dwell is always 0.250 seconds.*

**B** = Dwell (seconds) after lead-in move, before the first segment. Default = 0.060 seconds. The macro also uses the default if the call specifies **B0**.

**K** = Dwell (seconds) after contour. Default = 0.060 seconds. The macro also uses the default if the call specifies **K0**.

**Q** = Number of segments after lead-in. Range = 1 to 5, Default = 3

*Note: **G88** uses **Q** for the number of holes. When a program calls **G88** with **M2**, the number of segments is always 3.*

### Description

**G104** moves the cutting head to the pierce location in rapid traverse and calls **G84**. For rectangles, the lead-in ends at the center of the longer side. **G104** completes the lead-in, commands **M67** if the call specified **H1**, contours the shape with kerf compensation, and ends the cut with **M35**. **G104** then commands **M47** (unless called with **Z**-1) and returns.

## M2 LEAD-IN DESCRIPTION

The **M2** lead-in was developed to improve cutting in heavy steel plate. This method uses a “cross” lead-in, cutting a plus sign shape before the actual lead-in move. The size of the cross is one fourth of the lead-in length. The macro commands the cross moves at 7 IPM (178 mm/min).

After completing the cross shape, the macro commands a dwell before beginning the lead-in move. The macro call can specify this dwell with the **A** argument (except for **G88**).

The macro call can set the lead-in length with **W** and the lead-in angle with **V**. Lead-in feedrate is the contouring feedrate unless the macro call specifies a lower feedrate with **F**. The macro commands another dwell at the end of the lead-in, with a duration set with **B**.

Instead of beginning the shape at the contouring feedrate, the **M2** method first divides a portion of the first contouring entity into segments and commands them at increasing feedrates. The macro call can set the number of segments with **Q** (except for **G88**), and the length of each segment with **U**.

Each segment feedrate is a percentage of the contouring feedrate, based on the number of segments. The macro also maintains a minimum feedrate of 15 IPM.

The macro commands a third dwell at the end of the shape, to complete the return to the lead-in point. The macro call can set this dwell with **K**.

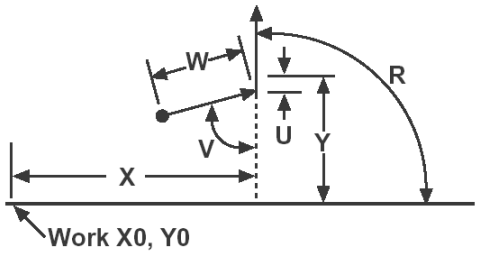
**M2** Lead-In Segment Feedrates (% of contouring rate):

| Q | Segment | | | | |
|---|---|---|---|---|---|
| | 1st | 2nd | 3rd | 4th | 5th |
| 1 | 50% | | | | |
| 2 | 40% | 80% | | | |
| 3 | 30% | 55% | 80% | | |
| 4 | 20% | 40% | 60% | 80% | |
| 5 | 20% | 35% | 50% | 65% | 80% |

## G105 LEAD-IN MACRO

**G105 X_ Y_ R_ (A_ B_ C_ F_ H_ M_ Q_ U_ V_ W_)** Lead-In Macro

**G105** provides the flexibility of a programmable lead-in to user-programmed contouring paths. The length, angle, speed, and type of lead-in can be changed by editing the **G105** macro call arguments at the machine.




**X, Y** = Work coordinates for end of lead-in.

**R** = Direction angle (in degrees) of the first contouring move after the lead-in. For **R0**, the lead-in is parallel to the local +x direction. The macro interprets positive **R** as counterclockwise.

**C** = Compensation direction. Default is **C41** (for **G41** = left side). **C42** is for **G42** = right side.

**F** = Feedrate for lead-in (see **G104**).

**H** = Optional pressure flag:

**H0** = Off (default)
**H1** = On. **G105** commands **M67** at the end of the lead-in when called with **H1**.

**M** = Lead-in type: (See **G104** descriptions)

**M0** = Single feedrate (default)
**M1** = Multi-step at increasing feedrates
**M2** = Cross lead-in followed by segments with increasing feedrates

**U** = Length of the last lead-in entity.

For **M0**, **M1**: **U** is the length of the last lead-in move which **G105** commands in line with the first contouring move. Default and minimum length is one kerf width.
For **M2**: **U** is the length of each contouring segment inserted after the lead-in. Default and minimum length is .080 inch. **G105** commands the **M2** segments in the same direction specified by **R**, with the last segment ending at the **G105** x and y coordinates.

**V** = Angle between lead-in line and first contouring move, in degrees.

Default **V** is 90 degrees (perpendicular lead-in). **G105** also use the default if called with **V0**. Use **V360** to command the lead-in in the same direction as the first contouring move. Positive angle is toward the beam side.

**W** = Lead-in length. Default length is 0.25 inches. For **M2**, minimum **W** is eight kerfs.

### Arguments Associated with M2 Only

**A** = Dwell (seconds) before lead-in move. Default = 0.250 seconds.

**B** = Dwell (seconds) after lead-in move. Default = 0.060 seconds.

**Q** = Number of segments after lead-in. Range = 1 to 5; default = 3

### G105 Description

**G105** calculates the starting position based on **R**, **U**, **V**, **W**, **X**, and **Y**. The macro moves the nozzle to that position and calls **G84** to start the cut. **G105** contours the lead-in with Kerf compensation and returns with the beam on.

## PROGRAM STRUCTURE

NC Programs are instructions for motion interpolation, machine functions and program control. NC Programs for the CINCINNATI INCORPORATED Laser System are ASCII text files written in the ISO format (International Standard Organization).

NC Programs execute from the program name down to the end of program statement. Each line is one NC block that may contain statements, expressions, program control, **G** code or **M** code commands. Each block can have a line number (also called the sequence number) assigned to it.

A program contains the following items:

- Program Name
- Program Body
- Optional Comments
- Program Body
- Optional Line Numbers
- Optional Block Delete Character (per line)
- End of Program (**M02** or **M30**), or End of Subprogram (**M99**)

### PROGRAM NAME

Program names are required for subprograms included in the same file with other (main or sub) programs. Otherwise, program names are optional. The block containing the program name can begin with either a colon character (:) or the letter **O**, followed by a string of up to 128 alphanumeric characters (see Notes). The end of the name is determined by a space.

*Note: There is one case where the program name cannot include alphabetic characters: The program name must be an integer number (maximum 9 digits) when the program is a subprogram called by a macro and the macro call specifies the subprogram name as a macro argument.*

### PROGRAM BODY

The program body contains one or more NC blocks. When the program is displayed as a text file, each block is one line of text. At the end of every block is the ISO character for line feed.

When the operator loads a program, the control checks the syntax of each block. If a block contains a syntax error, the control displays the message: “Incorrect syntax for laser applications”.

A block of code may consist of any of the following:

- NC move command (**G** code)
- NC modal command (**G** code or **Feedrate**)
- Math function (variable assignment)
- Logic statement (**IF** [ ... ] **THEN** ... , **GOTO** ... etc.)
- Machine (**M** code) function
- Macro call

Normally the control executes each block before proceeding to the next block. However, when a block assigns a math function to a variable, the control may look ahead and begin processing the math function during the execution of preceding blocks.

## BEAM ON AND OFF COMMANDS

To begin a laser cutting sequence, move the cutting nozzle to the starting location and command **G84** (or **G85**) in the next block. End the cut sequence with **M35** (or **M37** or **M135**).

*Example:*

**G89 P SS120N2.LIB**
**G00 G90 X10.5 Y6.25**
**G84** (BEAM ON)
**G41**
**G01 X10.75 F100**
**G03 X10.75 Y6.25 I-.5 J0 F300**
**M35** (BEAM OFF)
**G00 X11.5**

## PROGRAM COMMENTS

Because NC program statements are usually not direct readable text, it is sometimes convenient to add comments to the program. For example, the top of a program may contain set-up instructions for the operator.

Comments can be added to the program by enclosing the comment characters between parentheses ( ), or by starting the comment with an apostrophe ( ‘ ).

When the control reads a left parenthesis in a program block, it interprets all characters to the right as a comment, until it finds a right parenthesis. The block terminates with the end of block character (line feed). When the control reads an apostrophe in a program block, it interprets all characters to the right as a comment, until it finds the end of block character.

*Examples:*

**G92 X0 Y0** (SET WORK COORD.)
**M98 P2000** ‘ CALL PROGRAM 2000

## PROGRAM LINE NUMBERS

When a block directs execution to another block with a **GOTO** or **M99 P__** command, the destination block requires a line number. Otherwise, line numbers are optional. To assign a line number, begin the block with the character **N** followed by an integer. The line number has a usable range of 1 to 999999. The block may contain the block delete character before the line number.

*Example:*

**N2300 G01 X50**
(This example assigns line number 2300.)

The CNC Run Window displays line numbers with the program text.

## BLOCK DELETE

The operator can control the execution of a block with the **Beam ON** function. When a block begins with the / (forward slash) character, the control does not execute the block if Block Delete program mode is active (see the **PROGRAM MODES** topic). The operator can toggle this function on or off any time during the execution of the NC program.

## END OF PROGRAM

### M02

**M02** completes execution of a program. All previously requested M codes are reset and local variables are set to zero. However, the CNC Run Window does not return to the top of the program.

### M99 (P_)

The **M99** block completes execution of a subprogram. The CNC Run Window returns to the program that called the subprogram and displays the next block.

### M30

The **M30** block completes execution of a program. All previously requested **M** codes are reset, local variables are set to zero and the CNC Run Window returns to the top of the program.

## SUBPROGRAMS AND MACROS

Subprograms are separate CNC programs that execute when called by another program. The subprogram returns to the calling program when finished. The control maintains modal conditions established in the calling program unless the subprogram changes the modal condition.

Subprograms are useful for repeating a series of commands. For example, a main program can use a subprogram to repeat a part on a sheet.

There are two types of subprogram calls, **M98 P_** and **G65 P_**. The call specifies the subprogram name after **P**. If the subprogram is in the same file as the calling program, then the call only needs the program name after **P**. If the subprogram is in a separate file, the call should command **P** followed by the file name including its extension (if any) and its path if different from the calling program.

Each type of subprogram call has different properties and applications.

*Examples:*

**M98** P__ (L_)

**G65 P__ (L_) ( A_ B_ C_ D_** etc. **)**

A single program block can call a subprogram more than once, by commanding **M98** or **G65** with **L** followed by the number of times to repeat the subprogram.

*Example:*

**M98** P1000 (L3)

(This example calls a subprogram named “1000” three times before returning.)

## LOCAL VARIABLES

The major difference between **M98** and **G65** is the treatment of local variables. When a program calls a subprogram with **M98**, the two programs share the same set of local variables (#1 through #99). In other words, “local” variables become “common” variables between a program and a subprogram called with **M98**. The control assigns undefined local variables a value of zero.

When a program calls a subprogram with **G65**, the subprogram has its own set of local variables. The calling program can assign values to most of the variables #1 through #26 in the subprogram by including arguments in the **G65** call. This allows the calling program to pass data to the subprogram without assigning separate common variables.

Each argument is a letter followed by a numerical value for its corresponding local variable. The calling program cannot use arguments **G**, **L**, **N**, **O,** or **P**. The subprogram can use local variables that would correspond to **G**, **L**, **N**, **O,** and **P**, but the calling program cannot assign their values with arguments in the **G65** call.

Since program names in the CINCINNATI control can have alphabetic characters, the **G65** block requires a space between the last character of the program name and the first argument letter.

*Examples:*

**G65 PMYSUBA1**
(This example calls a subprogram named “MYSUBA1”.)

**G65 PMYSUB A1**
(This example calls subprogram “MYSUB” with argument A equal to 1.)

To be consistent with local variable assignments used by other CNC controls, the CINCINNATI control assigns arguments **D** through **K** to local variables out of sequence with their alphabetical order.

This table shows how the control assigns local variables to **G65** arguments:

| ARGUMENT | LOCAL VARIABLE |
|---|---|
| A | #1 |
| B | #2 |
| C | #3 |
| D | #7 |
| E | #8 |
| F | #9 |
| H | #11 |
| I | #4 |
| J | #5 |
| K | #6 |
| M | #13 |
| Q | #17 |
| R | #18 |
| S | #19 |
| T | #20 |
| U | #21 |
| V | #22 |
| W | #23 |
| X | #24 |
| Y | #25 |
| Z | #26 |

*Example:*

**G65 P2000 X12.5 Y3.5**
(This example calls a subprogram named “2000”. When the program named “2000” starts, it has local variables #24 = 12.5 and #25 = 3.5.)

The format for subprograms is the same as other programs, except subprograms use **M99** instead of **M30** for the End of Program statement. If a subprogram does not end with **M99**, the control displays the message: "S*ubprogram call without a return statement"*.

### NESTED SUB-PROGRAM CALLS

Subprograms can call other subprograms with **M98** or **G65**, until the total calls are nested 10 deep.

*Note: CINCINNATI Cutting Macros and Grid Macros are subprograms and contain subprogram calls. The total nesting limit available to the user is reduced when a program calls these macros.*

This table shows how CINCINNATI macro calls affect subprogram nesting:

| MACRO | EQUIVALENT SUBPROGRAM CALLS |
|---|---|
| G73 | 3 |
| G76 | 3 |
| G83 | 3 |
| G86 | 3 |
| G88 | 3 |
| G104 | 2 |
| G65 P9800 | 2 |
| G65 P9900 | 2 |

## PROGRAM VARIABLES

### LOCAL AND COMMON VARIABLES

Local and common variables can be displayed and edited by opening the *Local/Global Variables* window from *Variables* menu of the Laser System HMI .

**CAUTION**

**Editing variables while a program is running can cause improper execution and is strongly discouraged.**

*Note: The **Local/Global Variables** window will display variables with the current values they have in the control buffer. However, the buffer may be several blocks ahead of the currently executing NC program block.*

#### Local Variables: #1 - #99

Each program has 99 local variables named **#1** through **#99**. Each subprogram called with **G65** also has its own set of local variables. Subprograms called with **M98** share the same set of local variables with the calling program. All local variables are zero by default and return to zero after **M30** or Program Rewind.

#### Common Variables: #100 - #999

All programs and subprograms share a set of common variables named **#100** through **#999**. Common variables are not cleared by **M30**, pressing the "RESET" button or turning off control power.

One common variable can be reserved for use by CINCINNATI INCORPORATED. The General Configuration window has a 'Set Global Variable' check box in the Process Library Feedrate section. If that option is selected, **G89** writes the cutting feedrate parameter from the Process Parameter file into a common variable. The *Variable No.* configuration setting specifies the variable number (148 by default). **G89** sets the variable with a value consistent with the **G20** / **G21** (inch / metric) units mode in effect when the program calls **G89**. However, the variable does not change if the program changes **G20** / **G21** mode after calling **G89**.

## SYSTEM VARIABLES

System variables give the programmer the ability to read and write information for special functions in the CNC.

There are four types of system variables:

| SYSTEM VARIABLES | DESCRIPTION |
|---|---|
| #2000 - #2999 | Offset Data |
| #3000 - #3999 | CNC Data |
| #4000 - #4999 | Modal Data |
| #5000 - #5999 | Coordinate Data |

### Offset Data System Variables

### Kerf Offset: #2000

An NC program statement can read or write the cutter radius value used for kerf width compensation with **#2000**. The variable value is consistent with the active units mode (**G20** inch, **G21** mm).

An NC program can read the active compensation value by assigning the value of **#2000** to a program variable or using **#2000** in a math or logic statement. The program can change the active compensation value by calling **G89**, or by assigning a value to variable **#2000**.

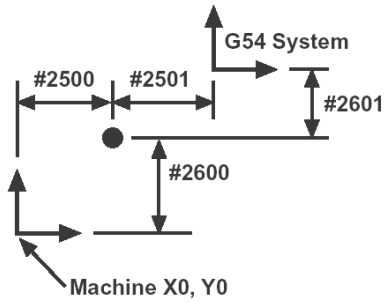
### Work Coordinate System Offsets

An NC program can read the value of a Work Coordinate Offset by assigning the value of its system variable to a program variable, or by using the system variable in a math or logic statement. The program can also change the Work Offsets by assigning values to the system variables. The control interprets the variables in the active units (**G20** inch, **G21** mm).

| SYSTEM VARIABLE | Work Offset Group | Axis |
|---|---|---|
| #2500 | External | X |
| #2600 | External | Y |
| #2501 | 1 (G54) | X |
| #2601 | 1 (G54) | Y |
| #2502 | 2 (G55) | X |
| #2602 | 2 (G55) | Y |
| #2503 | 3 (G56) | X |
| #2603 | 3 (G56) | Y |
| #2504 | 4 (G57) | X |
| #2604 | 4 (G57) | Y |
| #2505 | 5 (G58) | X |
| #2605 | 5 (G58) | Y |
| #2506 | 6 (G59) | X |
| #2606 | 6 (G59) | Y |

External work offsets (**#2500** and **#2600**) are normally zero. Programs use **#2500** and **#2600** to change the distance from machine **X**0, **Y**0 to a common reference point for the **G54** through **G59** work coordinate systems.

*Example:*




## CNC Data System Variables

### Clock Variables:

Date: **#3011** YYYYMMDD
Time: **#3012** HHMMSS
Day: **#3013** D

The control maintains these system variables for clock functions. An NC program statement can assign their value to a variable or use the system variables in a math or logic statement. A program cannot change these system variables by assigning a value to the variable.

### CNC Date: System Variable #3011

The CNC sets system variable **#3011** with the date and calendar year. The variable value is an eight-digit integer with the year, month and date separated as shown:

yyyymmdd

The year can be 0 to 9999, month is 01 to 12, and date is 01 to 31.

### CNC Clock: System Variable #3012

The CNC maintains **#3012** with the current time from the CNC clock. The variable value is a six-digit integer with the hour, minute and second separated as shown:

hhmmss

The data range for **#3012** is 000000 to 235959. The hour is in 24-hour clock format (Military Time), where 000001 is one second after midnight.

### Weekday: System Variable #3013

Programs can use **#3013** to read the day of the week. The variable value is a one-digit integer, 1 thorough 7 for Sunday through Saturday.

| Sun | Mon | Tue | Wed | Thu | Fri | Sat |
|---|---|---|---|---|---|---|
| 1 | 2 | 3 | 4 | 5 | 6 | 7 |

### Beam On Time: System Variable #3015

This variable accumulates the elapsed time between **Beam ON** (**G84** or **G85**) and **Beam OFF** commands (**M35**, **M37**, **M42**, **M47** or **M30**). The accumulated time includes any dwell times.

System Variable **#3015** is a floating-point number in seconds. Resolution is .001 sec.

The control resets **#3015** to zero each time the **CNC/HMI** application is started. An NC program cannot change variable **#3015**. The *Machine Statistics Window* (see **Statistics**) displays total accumulated *Beam ON time*.

To measure the beam-on time during a program, save **#3015** in a common variable and subtract the saved value from the new **#3015** value before the program ends.

*Example:*

**#100 = #3015** (START)
.
.
**#100 = #3015 – #100** (END)

### Modal Data System Variables

Programs can use the **#4000** series system variables in a math or logic statement. A program cannot change a **#4000** series variable by assigning a value to the system variable.

| SYSTEM VARIABLE | DESCRIPTION |
|---|---|
| #4001 | 0, 1, 2, or 3 for G0, G1, G2 or G3 |
| #4003 | 90 or 91 for G90 or G91 |
| #4006 | 20 or 21 for G20 or G21 |
| #4007 | 40, 41 or 42 for G40, G41 or G42 |
| #4008 | 54 through 59 for G54 through G59 |
| #4011 | 50 or 51 for G50 or G51 |
| #4015 | 61 or 64 for G61 or G64 |
| #4016 | 68 or 69 for G68 or G69 |
| #4109 | Modal Feedrate for G1, G2 and G3 |
| #4114 | Value following address N in the last executed block. If the last block did not specify N, then #4114= - 1. |

### Coordinate System Variables

A program can determine the machine or work coordinates of the last completed block by reading system variables. The control maintains the variables in the same units as the active units mode (**G20** inch, **G21** mm). The program cannot change these system variables by assigning a value to the variable.

*Machine Coordinates:*

**#5021** Machine x position
**#5022** Machine y position
**#5023** Machine z position
**#5061** Position Capture machine x position
**#5062** Position Capture machine y position

*Work Coordinates:*

**#5041** Work x position
**#5042** Work y position
**#5043** Work z position

# AUXILIARY FUNCTIONS

## MATH FUNCTIONS

NC programs can use the math functions listed in the following table to assign a calculated value to a variable, or to substitute a calculation for a numerical value. When processing program blocks containing math expressions, the CNC applies the following default order of precedence to the math operations:

1. Functions (SQRT, ABS, TAN, etc.)
2. Multiplication and division
3. Addition and subtraction

In the following examples, “a” represents a variable and “b” and “c” represent variables, constants or functions.

| FUNCTION | EXAMPLE |
|---|---|
| Assignment | a = b |
| Addition | a = b + c |
| Subtraction | a = b - c |
| Multiplication | a = b * c |
| Division | a = b / c |
| Binary Addition | a = b OR c |
| Binary Subtraction | a = b XOR c |
| Binary Multiplication | a = b AND c |
| Sine | a = SIN [ b ] |
| Cosine | a = COS [ b ] |
| Tangent | a = TAN [ b ] |
| Arc Tangent | a = ATAN [ b ] |
| Square Root | a = SQRT [ b ] |
| Rounding | a = ROUND [ b ] |
| Truncating | a = FIX [ b ] |
| Add 1 for fraction | a = FUP [ b ] |
| Absolute Value | a = ABS [ b ] |
| BCD to Binary | a = BIN [ b ] |
| Binary to BCD | a = BCD [ b ] |

**SIN [ ]** Sine of an angle in degrees.

**COS [ ]** Cosine of an angle in degrees.

**TAN [ ]** Tangent of an angle in degrees.

**ATAN [ ]** Arc-tangent of an expression.
The result of the ATAN function is in degrees, between -90 and +90.

**SQRT [ ]** Square root of a positive number.
Evaluating the square root of a negative number produces an error message.

**ROUND [ ]** Rounding off a number.
Expressions with fractional portions 0.5 and above are rounded to the next higher integer. Expressions with fractional portions less than 0.5 are rounded to their integer value.

**FIX [ ]** Truncating a number.
Any fractional portion is ignored. The number is reduced to its integer value only.

**FUP [ ]** Add 1 for fraction.
If the number has any fractional portion, the FUP function removes the fraction and adds 1 to the integer portion. If the number is already an integer (no fractional part), it stays the same.

**ABS [ ]** Absolute Value
ABS [ ] returns the absolute value of a variable or function.

**BIN [ ]** BCD to Binary number conversion
The BIN function converts the specified value from Binary Coded Decimal to Binary.

**BCD [ ]** Binary to BCD number conversion
The BCD function converts the specified value from Binary to Binary Coded Decimal.

### Braces [ ]

Math expressions can use braces to override the default order of precedence for math operators, and identify conditional expressions. There is no limit on the number of nested braces.

*Example 1:*

**#1** = 3 * [2 + 3]

The braces set the priority of the math functions; addition is performed before multiplication in Example 1.

*Example 2:*

**#2** = SIN[2 * [3 *[2 + 3]]]

In Example 2, the statement completes addition first. The sum is multiplied by 3 and that product is multiplied by 2. The SIN function operates on the product of the multiplication.

Each left brace must have a right brace. If the statement contains an error in brace syntax, the control displays the message "*Brace open/close error"*.

## LOGIC FUNCTIONS

Logic functions include conditional expressions and program control commands.

### Conditional Expressions

A program uses a conditional expression to compare the value of a variable, constant or calculation with another variable, constant or calculation. The program block can use the result of the comparison to direct program flow.

| COMPARISON | EXAMPLE |
|---|---|
| Equal | [b EQ c] |
| Not Equal | [b NE c] |
| Greater Than | [b GT c] |
| Less Than | [b LT c] |
| Greater Than or Equal | [b GE c] |
| Less Than or Equal | [b LE c] |

The CNC evaluates the conditional expression for proper syntax and a true or false condition. The evaluation of a conditional expression occurs from left to right.

## Program Control Commands

A program can determine which blocks the control executes by using the program control commands:

- GOTO
- IF [ ] GOTO
- IF [ ] THEN . . ELSE . . ENDIF
- WHILE [ ] . . END

### GOTO Statement

The control normally executes program blocks in sequential order (top to bottom through the program). A program can direct the control to execute any numbered block with the **GOTO nnn** command, where "nnn" is the sequence number of the destination block.

A program block specifies its sequence number with '**N**' followed by an integer number. To direct the control to a block with a sequence number, command **GOTO** followed by the numerical value of the sequence number. The **GOTO** command does not include the "**N**" character.

*Example:*

```
N020 GOTO 50
N030 . . .
N040 . . .
N050 G01 X10 Y5
```

*(In this example, the control executes the block with sequence number **N050** immediately after the block with sequence number **N020**.)*

### IF [ ] GOTO Statement

**IF** [<conditional expression>] **GOTO** <line>

If the expression is true, the program jumps to the specified line number. If false, the program continues with the next block.

### IF [ ] THEN … ENDIF Statement

**IF** [<conditional expression>]
**THEN** < expression >
**ENDIF**

The CNC evaluates the conditional expression. If true, the control executes the block containing **THEN**. If the expression is false, the program jumps to the **ENDIF** block.

The **THEN** block must be a separate block immediately following the **IF** block. Each **IF THEN** pair must be followed by a separate **ENDIF** block. **IF THEN** statements can be nested four deep.

*Example:*

```
IF [#1 EQ #2]
THEN #3 = #4 / #5
ENDIF
```

### IF [ ] THEN … ELSE … ENDIF Statement

**IF [**<conditional expression>**]**
**THEN** <expression>
**ELSE** <expression>
**ENDIF**

If the conditional expression is true, the control executes the block containing **THEN** and the program jumps to the **ENDIF** block. If the expression is false, the control executes the block containing **ELSE**.

The **THEN** block must be a separate block immediately following the **IF** block. The **ELSE** and **ENDIF** blocks must also be separate blocks. **IF..THEN..ELSE..ENDIF** statements can be nested four deep.

### WHILE Statements

The control accepts two types of **WHILE** statements:

**WHILE** . . **END** Statement:

**WHILE [**<conditional expression>**] DO m**
.
.
**ENDm**

(In the **WHILE** . . **END** structure, "**m**" is an integer from 1 to 3, used to identify nested loops.)

**WHILE . . ENDWHILE** Statement:

**WHILE** [<conditional expression>]
.
.
**ENDWHILE** (one word)

The control evaluates the conditional expression. If true, the control executes the block(s) between **WHILE** and **END** (or **ENDWHILE**), then the program returns to the **WHILE** block and repeats the process. When the expression becomes false, the program jumps to the block after **END** (or **ENDWHILE**).

**WHILE** loops can be nested up to five deep. To avoid the possibility of an infinite loop, a block between **WHILE** and **END** normally changes the status of the conditional expression.

*Example:*

```
#1=0
WHILE [#1 LE 50] DO2
.
.
#1=#1+1
END2
```

The blocks between **WHILE** and **END** may include a **GOTO** command to exit the loop.

## AUXILIARY COMMANDS

### DPRNT Command (Option)

The program can transmit a text string to an RS-232 port using the **DPRNT** command if the Laser System has the *CNC Serial Output* option. Serial COM port settings for the optional **DPRNT** function are configured in the *Serial Port Configuration* window.

### DPRNT text string

The text string can begin immediately after **DPRNT**, or the block may include spaces between **DPRNT** and the first string character. The control reads the text string until it finds a space or end-of-block character. The text string cannot include braces [ ]. If the text string includes a variable name, the control will only transmit the variable name (the control will not transmit the value of the variable).

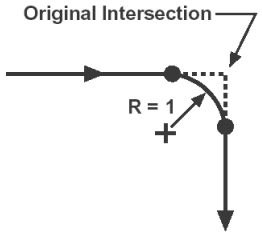
When the control processes the **DPRNT** statement, it adds the **DC2**, nulls, and **DC4** commands, and transmits the string. The control does not use **OPEN** or **CLOSE** commands for **DPRNT**.

### Automatic Corner Rounding

A program can command a rounded corner by specifying the intersection point of the two side elements and the corner radius. The control inserts a circular arc tangent to any two contouring moves when the first move ends with a comma followed by **R** and the arc radius. The first and second contouring moves can be **G01**, **G02,** or **G03**.

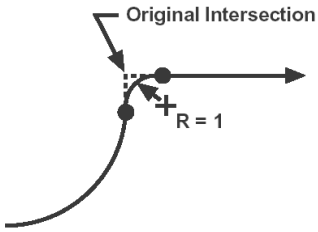
*Example:*

**(G90) G01 X4 Y2, R1**
**G01 Y0**




*Example:*

**(G90) G03 X5 Y5 R5, R1**
**G01 X10 Y5**

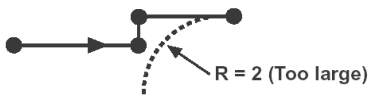



If a contouring block (**G01**, **G02**, or **G03**) does not follow the block specifying corner rounding, the control displays the message: "*Illegal corner rounding command*".

If the control cannot insert the programmed arc tangent to both original lines (or arcs), the control displays the message: "*Corner radius too large*".

*Example:*

**(G90) G01 X5 Y0**
**G01 X5 Y1, R2**
**G01 X8 Y1**




If the angle between the tangent lines is 1 degree or less, the control ignores the corner rounding request and contours the original intersection.

## WORKPIECE EDGE DETECTION

Programs use this option to find the location and orientation of a sheet of material that was not placed exactly at the machine origin (X0, Y0) or parallel with the machine axes. It uses the noncontact head to detect the coordinates of the material edges. The NC control uses the edge coordinates to automatically shift and rotate the work coordinate system, effectively aligning the part program with the workpiece. This method allows full utilization of the sheet, regardless of minor variations in material position on the pallet. Users can apply Workpiece Edge Detection with an automated pallet loading system or in cases where the material is not clamped. The noncontact head is the only hardware required to use this option.

### How It Works

The control finds the material position and orientation by detecting the edge of the sheet at three points. To find the location of an edge, the control first commands the nozzle to a low standoff (called the scan standoff) and locks the Z-axis servomotor. The control then changes the target standoff to a higher value called the detection threshold. The commanded servo lock prevents the Z-axis from moving up to the higher standoff. The control moves the cutting head horizontally in the direction of the expected edge until the measured standoff appears to increase to the detection threshold. This change in the apparent standoff occurs when the nozzle reaches the edge. The control records the X-axis and Y-axis positions when it detects the edge. After adjusting for slight offsets between the detection point and the actual edge, the scan is complete.

### Setup

If the Laser System has never used Workpiece Edge Detection, complete the following steps:

1. Open the Maintenance Configuration window, select the Options tab and verify that the check box for 'Workpiece Edge Detection' is checked.
2. Set the following common variables:
   #520 = 0.5
   #521 = 7.0
   #529 = 25.4

### Calibration

1. Position a 9 x 9 inch (230 x 230mm) or larger scrap workpiece on the worktable. Mild steel in any thickness from 16 gage to 10 gage works well.
2. Perform Standoff Calibration (see Section 7 of the Laser System **Operation, Safety, and Maintenance Manual**).
3. Jog the cutting head to the approximate center of the workpiece and run NC program **"***cal_edge_det.cnc*"

   This program file is in folder *"C:\ CNCLSR32 \ PROGRAMS \ UTILITIES"*
   Calibration function for Edge Detection calculates scanning offsets for noncontact head.

   *Note: Use at least a 9" x 9" piece of material and use calipers to verify cut square is 3.00 x 3.00 ± .003 inches.*

```
G90 G20
G89 P MS060N2.LIB
(Edit G89 block for appropriate library file name)
M47
G69
#11=#5021
#12=#5022
G92 X#5021 Y#5022
(G76 cuts a 3" x 3" square)
G76 X#5021 Y#5022 I3 J3 C.1 R0
M0 (Remove square, press "CYCLE START")
G65 P9700 A0.020 B0.070 C3 D3 F60 X#11 Y#12
M30
```

4. This program cuts a 3” x 3” (76.2 x 76.2mm) square hole in the workpiece. Be sure that the **G89** block specifies the correct library file name for the material.

5. When the hole is cut and the program stops, remove the square slug from the hole. Be sure not to bump the workpiece in which the hole was cut. Any movement of the test hole before the calibration scan (Step 6) will cause an equal size error in the **Edge Detection** function. (If the workpiece is bumped, the program must be restarted from the beginning.)

6. After removing the square slug, press “CYCLE START”. The calibration program will then scan the square hole to set up its internal calibration values.

7. When the calibration is complete, measure the dimensions of the 3” x 3” (76.2 x 76.2mm) square hole (not the slug). If the hole dimensions differ from the programmed value by more than 0.002” (0.05mm), correct the kerf value in the material library and/or adjust the focus, then repeat the calibration. Increase the kerf parameter if the hole is too large. Any error in the size of the test hole will cause an equal size error in the **Edge Detection** function.

## Operation

1. Position the workpiece near the “home” corner of the worktable with the longer side of the workpiece along the X-axis, that is, lengthwise on the worktable. The edges of the workpiece can be anywhere from zero to +2.0 inches (50.8mm) from the “zero” position of the worktable along the X-axis and along the Y-axis.

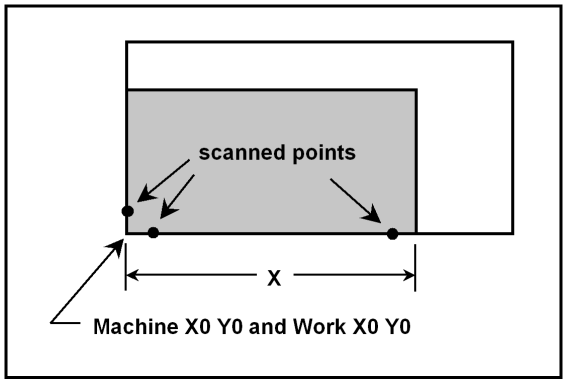
   *Note: All programmed moves must be within the maximum cutting area of the worktable. If the program attempts a move beyond the machine limits, the program will stop and generate an overtravel alarm.*

2. Insert the following command in the NC program, after a **G89** command and before any move commands:

   **G65 P9712 Xn**

   In this command, “n” is the length of the workpiece in the X-axis direction.

   When the control executes this command, the cutting head will scan one workpiece edge at two points and a perpendicular edge at one point to determine the position of the two edges. The control will then shift and rotate the work coordinate system to make the X- and Y-axes coincide with the measured edges. The **Edge Detection** function ends when the control moves the cutting head to the X0, Y0 corner of the workpiece, in the partial Z-up (**M47**) position.

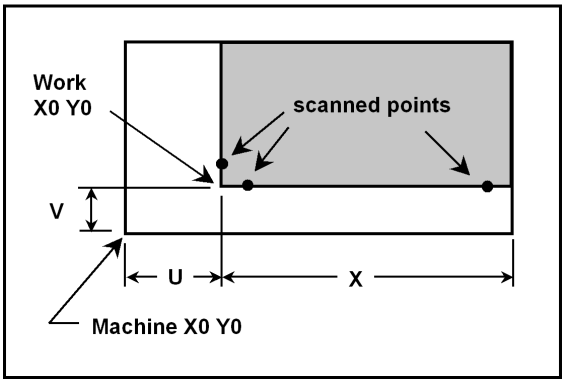



***Edge Detection using default U0, V0, R0***

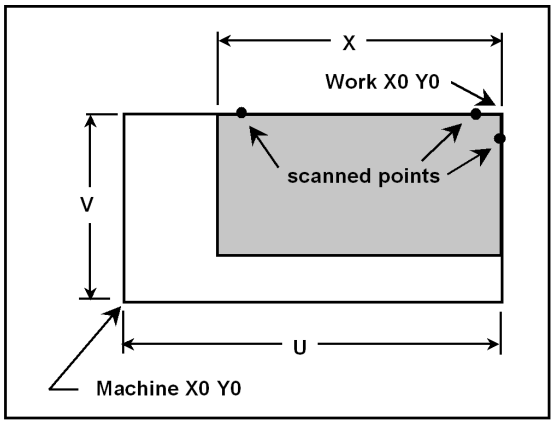
## Optional Functions

When the NC program calls the edge detection macro with only the X argument, the macro finds a workpiece with one corner near the machine X0 Y0 location. The NC program can also use the edge detection macro when the workpiece does not have a corner near machine X0 Y0. To locate a sheet with a corner near machine coordinates other than X0 Y0, the macro call includes the optional arguments **U** and **V**. The macro interprets **U** and **V** as machine coordinates of the sheet corner where the scanned edges intersect.

When the macro call does not specify the optional **R180** argument, the macro locates a sheet with the X and Y work coordinate axes in the same general direction as the machine axes. If the macro call specifies **R180**, the macro locates a sheet with the X and Y work coordinate axes in the opposite directions from the machine axes. The following figures show workpiece orientations that would use the optional **R**, **U**, and **V** arguments:




***Edge Detection using U, V, and default R0***




***Edge Detection using U, V and R180***

## Comments

The **Edge Detection** function works in a program using either English units (**G20**) or metric units (**G21**).

The **Edge Detection** function changes the commanded feedrate. After completing each scan, the control resets the commanded feedrate to the value of variable **#148.** If variable **#148** was not defined before the program called the 9712 macro, the macro will return with the commanded feedrate equal to the scan feedrate. If an error occurs, the 9712 macro might not properly reset the original feedrate.

The **Edge Detection** function changes the pierce and cut standoff setting. To ensure that the program uses the proper standoffs for cutting, always include a **G89** or **G102** command after the edge detection macro call.

*Example:*

```
G89 PMS060N2.lib
G65 P9712 X120.0
G89 PMS060N2.lib
```

To maintain the shifted coordinate system, the user program may not contain any of the following commands after calling the **Edge Detection** function: **G52**, **G53**, **G54** through **G59**, **G68**, **G69** or **G92**.

After the initial setup and calibration, be sure not to change common variable **#529**. This variable indicates the current system of units (inch or metric) for the stored calibration values. If this variable changes, repeat the initial setup steps and the calibration procedure.

To change the feedrate used when scanning an edge, modify the feedrate used in the calibration routine. The **F** argument in the **G65 P**9700 macro call (calibration) sets the scan feedrate.

*Note: Higher scan feedrates will reduce accuracy.*

## Specifications

| EDGE DETECTION SPECIFICATIONS | |
|---|---|
| Position Accuracy | ±0.100 inch (2.54 mm) |
| Sheet Position Tolerance | 0 to +2.0 inches (50.8 mm) from machine coordinate zero at the scan points |
| Scan Feedrate | Maximum programmable = maximum contouring feedrate.<br>Maximum practical = 100 IPM (2540 mm/min). |
| Scan Standoff | Minimum programmable = 0.010 inch (.254 mm).<br>Minimum practical = 0.020 inch (0.508 mm).<br>Maximum programmable = 0.400 inch (10.2 mm).<br>Maximum practical = 0.100 inch (2.54 mm). |
| Detection Threshold | Minimum = scan standoff.<br>Maximum= 0.400 inch (10.2 mm).<br>Maximum practical = 0.250 inch (6.35 mm). |

## Troubleshooting

*Note: Error messages shown in quotes in this table will appear in the CNC Run Window, at the line where the macro program stopped after finding the error.*

| PROBLEM | POSSIBLE CAUSES | SOLUTION |
|---|---|---|
| "Standoff out-of-range" | The macro 9700 (Calibrate Sensor) scan standoff argument (A) is outside the acceptable range of 0.010 inches to 0.400 inches. | Correct the "A" argument in the macro call. |
| "Threshold out-of-range" | The macro 9700 (Calibrate Sensor) threshold argument is less than the scan standoff argument or greater than 0.400 inches.<br>• The "A argument, the "B" argument, or both are incorrect when calling macro 9700. | Correct the "A" and/or "B" argument in the macro call. |
| "Move distance too small" | Macro 9701 (Scan) move distance is less than the minimum value set by common variable #520.<br>• Common variable #520 is incorrect. | Set common variable #520 to 0.500 inches (12.7 mm). |
| "Move distance too large" | The macro 9701 (Scan) move distance is greater than the maximum value set by common variable #521.<br>• Common variable #521 is incorrect. | Set common variable #521 to 7.000 inches (177.8 mm). |
| "Started too close to edge" | Macro 9701 (Scan) detected the edge within 0.250 inches (6.35 mm) of the scan starting point. If the start point is too close to the edge, the macro does not maintain detection accuracy.<br>• The edge of the workpiece is too close to the start point. | Move the workpiece. |
| "Edge not detected" | Macro 9701 completed the scan, but did not detect an edge.<br>• The edge of the workpiece is too far from the start point. | Move the workpiece. |
| "X-axis length too small" | The "X" argument in the 9712 (Align to Sheet) macro call is less than 12 inches (304.8 mm). It is not practical to scan a workpiece less than 12 inches long. | Correct the "X" argument in the macro call. |
| "Arguments missing" | A block calling macro 9700 (Calibrate Sensor) or macro 9712 (Align to Sheet) does not have the required arguments. | Make sure the arguments in the macro call match the required arguments. See the macro description for the argument list. |
| Laser-cut shapes are not accurately positioned relative to the workpiece. | • The program commanded G69 after calling the Align to Sheet macro.<br>• The program commanded G92 after calling the Align to Sheet macro.<br>• The nozzle tip was replaced or damaged since the last calibration.<br>• The cutting head was removed and replaced since the last calibration.<br>• The scan feedrate is too fast. | • Modify the program.<br>• Modify the program.<br>• Repeat the calibration.<br>• Repeat the calibration.<br>• Repeat the calibration with a lower scan feedrate. |
| The cutting head missed the workpiece during a scan. | A scanned workpiece edge is more than 2.0 inches from the Machine X=0 or Y=0 position. | Move the workpiece. |

## Technical Information

This section includes additional technical information not needed for normal operation of the **Workpiece Edge Detection** function. This information is provided as a service aid to help debug problems that might occur in special applications. This section includes listings of the common variables and system variables used by the edge detection macros. In addition, this section includes the following information for each macro: a brief description of the macro, a list of arguments needed when calling the macro, a list of local variables used by the macro, the common variables returned by the macro, and a list of error messages generated by the macro.

### Common variables used by the workpiece edge detection macros:

**#131** = Xde1 = X-work coordinate of detection point adjusted for probe offset, but not detection shift

**#132** = Yde1 = Y-work coordinate of detection point adjusted for probe offset, but not detection shift

**#519** = Qr = Calculated coordinate system rotation angle (deg)

**#520** = Rprtg,min = Minimum scan distance from start position to target position [0.5 in]

**#521** = Rprtg, max = Maximum scan distance from start position to target position [7.0 in]

**#522** = Fcal = Feedrate at last calibration

**#523** = Rds = Detection shift at last calibration (positive indicates detection point is past actual edge)

**#524** = Xof = X-axis probe offset at last calibration

**#525** = Yof = Y-axis probe offset at last calibration

*Note: The probe offset is the distance from the effective probe center to the effective laser beam center.*

**#526** = Zsc = Scan standoff at last calibration

**#527** = Zth = Detection threshold at last calibration

**#529** = Units constant (1.0 for mm, 25.4 for inch)

**#530** = Xsh = Calculated X-axis translation

**#531** = Ysh = Calculated Y-axis translation

### System variables used by the workpiece edge detection macros:

**#5021** = Present X-machine coordinate

**#5022** = Present Y-machine coordinate

**#5041** = Present X-work coordinate

**#5042** = Present Y-work coordinate

**#5061** = X-work coordinate when SKIP signal received (edge detected)

**#5062** = Y-work coordinate when SKIP signal received (edge detected)

### Macro Descriptions

### Macro 9700 (Calibrate Sensor)

**Description:**

The macro call must specify the scan standoff and detection threshold values, Zsc and Zth, to be used for scanning with the noncontact head. A square calibration hole (at least 3” x 3”) must be cut prior to calling this macro. The macro call must specify the programmed coordinates Xcn and Ycn of the center of the hole and the measured dimensions Lx and Ly of the hole. The macro scans the calibration hole to determine the detection shift Rds and probe offsets Xof and Yof.

If an argument is missing or out of range, the macro preserves the results from the last calibration. If an error occurs after the calibration scan has started, the detection shift and probe offsets are set to zero.

**Arguments:**

**A (#1)** = Zsc = Scan standoff

**B (#2)** = Zth = Detection threshold

**C (#3)** = Dx = Measured edge-to-edge distance in X direction

**D (#7)** = Dy = Measured edge-to-edge distance in Y direction

**F (#9)** = Fr = Feedrate for calibration

**X (#24)** = Xcn = Nominal X-work coordinate of calibration hole center

**Y (#25)** = Ycn = Nominal Y-work coordinate of calibration hole center

**Local Variables:**

**#17** = Xds = X-axis detection shift

**#18** = Yds = Y-axis detection shift

**#19** = Xlm = Measured X-coordinate of left edge

**#20** = Xrm = Measured X-coordinate of right edge

**#21** = Ybm = Measured Y-coordinate of bottom edge

**#22** = Ytm = Measured Y-coordinate of top edge

**Returns:**

**#522** = Fcal = Feedrate at last calibration

**#523** = Rds = Detection shift at last calibration

**#524** = Xof = X-axis probe offset at last calibration

**#525** = Yof = Y-axis probe offset at last calibration

**#526** = Zsc = Scan standoff at last calibration

**#527** = Zth = Detection threshold at last calibration

**Calls:**

9701 (Scan)
9709 (Convert Units)

**Called by:** User program

**Error messages:**

"Standoff out of range"
"Threshold out of range"
"Arguments missing"

### Macro 9701 (Scan Subprogram)

**Description:**

This subprogram lowers the Z-axis to the scan standoff, puts the Z-axis in servo-hold and commands standoff equal to the detection threshold. It starts at the present position (Xpr, Ypr) and moves on a straight line path toward the specified target position (Xtg, Ytg) at the feedrate used in the last calibration. The macro uses **G31** to determine the position where the noncontact head detected an edge. It stops just beyond the detect position (or at the target position if it does not detect an edge) with a partial **Z-up** command. The subprogram does not return to the start position.

The subprogram modifies the pierce standoff and cut standoff from the values set by the most recent **G89** or **G102** macro call. The calling program must command a new **G89** or **G102** block immediately after calling the **9700** or **9712** macro. The subprogram returns with the commanded feedrate equal to variable **#148**. If variable **#148** was not defined before the macro call, the subprogram returns with the commanded feedrate equal to the scan feedrate.

**Arguments:**

**X (#24)** = Xtg = X-work coordinate of target

**Y (#25)** = Ytg = Y-work coordinate of target

**Local Variables:**

**#1** = Present X-work coordinate = starting X-coordinate

**#2** = Present Y-work coordinate = starting Y-coordinate

**#18** = Xprde = X-coordinate distance from start to detect

**#19** = Yprde = Y-coordinate distance from start to detect

**#20** = Rprde = Radial distance from start to detect

**#21** = Xdetg = X-coordinate distance from detect location to target

**#22** = Ydetg = Y-coordinate distance from detect location to target

**#23** = Rdetg = Radial distance from detect location to target

**#26** = Xprtg = Xpr - Xtg = X-coordinate distance from start to target

**#27** = Yprtg = Ypr - Ytg = Y-coordinate distance from start to target

**#28** = Rprtg = Radial distance from start to target

**Returns:**

**#131** = Xde1 = X-work coordinate of detection point adjusted for probe offset, but not detection shift

**#132** = Yde1 = Y-work coordinate of detection point adjusted for probe offset, but not detection shift

**Calls:** None

**Called by:**

9700 (Calibrate Sensor)
9702 (Find Hole)
9712 (Align to Sheet)

**Error messages:**
"Move distance too small"
"Move distance too large"
"Started too close to edge"
"Edge not detected"

### Macro 9709 (Convert Units)

**Description:**

If the active CNC units (inch or mm) are not the same as the edge detection parameter units, the 9709 macro converts the edge detection parameters to the active units.

**Arguments:** None

**Local Variables:** None

**Returns:** Parameter units consistent with CNC units.

**Calls:** None

**Called by:**

9700 (Calibrate Sensor)
9710 (Align to Holes)
9712 (Align to Sheet)

**Error messages:** None

## Macro 9712 (Align To Sheet)

**Description:**

This macro shifts and rotates the work coordinate system to align it with the sheet position.

**Arguments:**

**X (#24)** = Lx = Length of sheet along X-axis

**Local Variables:**

**#10** = Lx, min = Minimum sheet length [12 in]
**#11** = Lb = Backup distance from corner to test points [2.250 in]
**#12** = Rprtg = Distance from start to target when scanning an edge [2.500 in]
**#13** = #507 = Save value of pierce time so it can be restored
**#14** = Xa = X-work coordinate of sheet edge at position A
**#15** = Yb = Y-work coordinate of sheet edge at position B
**#16** = Yc = Y-work coordinate of sheet edge at position C
**#17** = Qr = Angle of sheet edge relative to X-axis based on Yb and Yc
**#18** = Xe = X-coordinate of home corner of sheet based on Xa, Yb, and Yc
**#19** = Ye = Y-coordinate of home corner of sheet based on Xa, Yb, and Yc

**Returns:**

**#519** = Qr = Calculated coordinate system rotation angle
**#530** = Xsh = Calculated X-axis translation
**#531** = Ysh = Calculated Y-axis translation

Coordinate system shifted and rotated.

**Calls:**

9701 (Scan)
9709 (Convert Units)

**Called by:** User program

**Error messages:**

"X axis length too small"
"Arguments missing"

# NC CODE LIST

| M CODE LIST | |
|---|---|
| **M00** | Program Stop<br>Cycle operation is stopped. All existing modal information remains unchanged. Cycle operation is resumed by depressing “CYCLE START”. |
| **M01** | Optional Stop<br>Same as M00 except code is ignored unless Optional Stop Mode is active. |
| **M02** | End of Program<br>End of main program. Click “REWIND” or load a new program to continue. |
| **M30** | End of program / Rewind<br>End of main program with rewind. Press “CYCLE START” to repeat entire program. |
| **M35** | Beam and gas off |
| **M36** | Z Hold mode<br>Nozzle height is locked in position, regardless of material height variation. Only recommended for short X/Y moves. |
| **M37** | Sequence off<br>Beam off, assist gas off, shutter close. |
| **M38** | Z Hold mode, timed<br>Nozzle height is locked in position, regardless of material height variation. M38P<value> determines length of time until nozzle resumes tracking material. <Value> is expressed is seconds times 1000. |
| **M41** | Z-down<br>Z-axis (nozzle) is commanded to travel down until it acquires its programmed standoff on either the upper or the lower pallet. |
| **M42** | Z-up Home<br>Z-axis (nozzle) is commanded to travel upward to its home position. Shutter is commanded to close. |
| **M43** | Lower Pallet Special Function enabled<br>Box / tube cutting mode. Z-axis (nozzle) begins tracking immediately, resulting in a slower down move; however, a greater variety of cutting heights is permitted. All pallet moves are disabled while this code is active. |
| **M44** | Lower Pallet Special Function disabled<br>Box / tube cutting mode canceled. |
| **M45** | Optional standoff mode<br>Z-axis commanded standoff is changed to the Optional (M45) Standoff setting found in the active Process Parameter. If head is not already tracking material, an M41 or G84 will still be necessary to command the head down. |
| **M47** | Z-up Partial<br>Z-axis is commanded to travel upward 0.80 inches (typ) according to setting in the *Z Setup* window. Use M47P<value> to customize the up distance on individual features (<Value> = inches times 1000). This command is useful when Z-axis needs to be raised somewhat prior to a rapid traverse move. |
| **M48** | Feedrate Override Inhibit<br>Feedrate override is fixed at 100% regardless of the position of the feedrate override switch. This function is canceled by M49, M30 or rewind |
| **M49** | Feedrate Override Enable<br>Feedrate override switch is active. This function cancels M48. |
| **M50** | Switch Pallets<br>Pallets are commanded to swap positions. One pallet must be in the cut position and one pallet must be in the load position for this command to be initiated. |
| **M51** | Auxiliary Timed Output On<br>Auxiliary relay is energized for length of time specified by the ‘P’ variable. The ‘P’ argument is expressed in milliseconds. |
| **M67** | Optional Assist Gas Pressure<br>The optional assist gas pressure defined by the G102 ‘I’ variable or defined in the Process Parameter file is commanded. |
| **M90** | Ball Transfer UP<br>Ball transfer mechanism is commanded to the UP position. This feature is typically used in conjunction with automated material handling systems, such as the CINCINNATI MMHS. |
| **M91** | Ball Transfer DOWN<br>Ball transfer mechanism is commanded to the DOWN position. This feature is typically used in conjunction with automated material handling systems, such as the CINCINNATI MMHS. |
| **M98** | Subprogram call<br>This code is used to call a subprogram. |
| **M99** | End of Subprogram<br>This code indicates the end of a subprogram and returns control to the main program. |

| M CODE LIST | |
|---|---|
| **M130** | Antidive disabled mode<br>Disables the automatic **Antidive** feature, which can help avoid damage to the cutting head due to programming errors. Thinner materials, where excessive material flutter is present when not well-supported, can falsely trigger Antidive in some applications. Note that proper use of the CINCINNATI Programming and Nesting software will ensure that the cutting head does not pass over a previously cut feature. |
| **M131** | Antidive enable<br>Re-enables the **Antidive** feature. M30 will also re-enable Antidive. |
| **M135** | Beam off<br>"Speed gas" **Beam OFF** feature. Assist gas stays on to save time on next sequence. |
| **M151** | Auxiliary output with confirmation<br>When an M151 code is commanded in an NC program, program execution pauses and waits for the corresponding M151 customer input to turn on. |

| G CODE LIST | |
|---|---|
| **G00** | Positioning-Rapid Traverse<br>X/Y axes are commanded to move at top speed to the end point as specified by G0X<value>Y<value>.<br>NOTE: Do not use G0 for cutting. |
| **G01** | Linear Interpolation<br>X/Y axes are commanded to move at programmed feedrate to the end point as specified by G01X<value>Y<value>. |
| **G02** | Circular Interpolation CW<br>X/Y axes are commanded to move at programmed feedrate, in a clockwise arc, to the end point as specified by G02X<value>Y<value>i<value>j<value>. |
| **G03** | Circular Interpolation CCW<br>X/Y axes are commanded to move at programmed feedrate, in a counterclockwise arc, to the end point as specified by G02X<value>Y<value>i<value>j<value>. |
| **G04** | Dwell<br>X/Y axes are commanded to a stop for a specified time duration. G04P<value> where <value> is expressed in milliseconds. |
| **G09** | Exact Stop<br>X/Y axes are commanded to a brief stop at end point. For rectangular moves, this will ensure a sharp corner. |
| **G20** | Inch mode<br>Inch mode is specified. All arguments, including X/Y arguments, are expressed in inches. |
| **G21** | Metric mode<br>Metric mode is specified. All arguments, including X/Y arguments, are expressed in millimeters. |
| **G31** | Position Capture move<br>X/Y axes are commanded to move as specified by X, Y and F arguments. If *Position Capture* input is activated during the move, X-axis and Y-axis positions at that time are recorded for use in coordinate scaling functions. |
| **G40** | Kerf Width Compensation Cancel<br>Kerf width compensation is canceled. |
| **G41** | Kerf Width Compensation Left<br>Kerf width compensation is activated, offsetting the beam path to the LEFT of the programmed path, according to the Kerf Width parameter found in the active Process Parameters. |
| **G42** | Cutter Radius Compensation Right<br>Kerf width compensation is activated, offsetting the beam path to the RIGHT of the programmed path, according to the Kerf Width parameter found in the active Process Parameters. |
| **G50** | Scaling Cancel<br>Scaling mode is canceled. |
| **G51** | Scaling<br>Scaling mode is enabled. |
| **G52** | Local Coordinate System Setting |
| **G53** | Machine Coordinate System Selection<br>X/Y axes are commanded to move to the ACTUAL machine position specified in X<value>Y<value>. |
| **G54** | Work Coordinate System 1 Selection<br>Work coordinate system #1 is specified. |
| **G55** | Work Coordinate System 2 Selection<br>Work coordinate system #2 is specified. |

| G CODE LIST | |
|---|---|
| **G56** | Work Coordinate System 3 Selection<br>Work coordinate system #3 is specified. |
| **G57** | Work Coordinate System 4 Selection<br>Work coordinate system #4 is specified. |
| **G58** | Work Coordinate System 5 Selection<br>Work coordinate system #5 is specified. |
| **G59** | Work Coordinate System 6 Selection<br>Work coordinate system #6 is specified. |
| **G61** | Exact Stop Mode<br>All subsequent X/Y axes are commanded to a brief stop at end point. For rectangular moves, this will ensure a sharp corner. |
| **G62** | Automatic Corner Override Mode |
| **G64** | Cutting Mode (Cancel G61 and G62) |
| **G65** | Macro Call |
| **G68** | Coordinate System Rotation<br>Coordinate system rotation is enabled. |
| **G69** | Coordinate System Rotation Cancel |
| **G84** | Begin pierce/cut sequence<br>Begin a new pierce and cut sequence, automatically commanding the cutting head to the desired standoff, the assist gas to the desired pressure, and the focus lens to the desired focus, per the specified Process Parameters. Optional 'T' codes can be used to enable enhanced operation: G84 T2 - Rapid Pierce; G84 T3 - No pierce (same as G85); G84 T4 - Pierce Only; G84 T5 - Rapid Pierce Only. |
| **G85** | Begin cut sequence<br>Begin cut sequence , Pierce Time = 0 (Same as G84 T3). |
| **G89** | Load new Process Parameters<br>G89 P <file name>.lib is used to load the active cutting/Process Parameters for a given material. |
| **G90** | Absolute Command<br>All subsequent X/Y arguments will be interpreted as ACTUAL desired positions / endpoints in the active coordinate system. |
| **G91** | Incremental Command<br>All subsequent X/Y arguments will be interpreted as RELATIVE to the present position. |
| **G92** | Work Coordinates Change |
| **G102** | Set Additional Process Parameters |
| **G103** | Pierce Ramping Power Control |
| **G120** | Disable Smart Rapids / Non-stop cutting<br>Disables special non-stop cutting mode. |
| **G121** | Enable Smart Rapids / Non-stop cutting<br>Enables non-stop cutting of thin gage material. This feature utilizes the speed gas feature, as well as advanced lead-in/lead-out trajectories, to maximize throughput. No pierce delay is permitted while in this mode. |
| **G125** | Auto-blend enabled (default mode)<br>Enables automatic calculation of maximum allowable machine acceleration. Feedrates may be reduced on some arc and corner moves in order to maintain reasonable accuracy and prevent excessive acceleration/deceleration. |

CINCINNATI®

COPYRIGHT © 2010
CINCINNATI INCORPORATED

CINCINNATI is a registered trademark of Cincinnati Incorporated.